Inventor®
and Its Applications

Third Edition

2010

David P. Madsen

Inventor
and Its Applications

by

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2010

Introduction

Inventor and Its Applications is a textbook providing complete instruction in mastering fundamental Autodesk Inventor® tools and modeling techniques. Typical applications of Inventor are presented with basic modeling and design concepts. The topics are covered in an easy-to-understand sequence and progress in a way that allows you to become comfortable with the tools as your knowledge builds from one chapter to the next. In addition, *Inventor and Its Applications* offers the following features:

- Step-by-step use of Inventor tools.
- In-depth explanations of how and why tools function as they do.
- Extensive use of font changes to specify certain meanings.
- Examples and discussions of industry practices and standards.
- Actual screen captures of Inventor features and functions.
- Professional Tips explaining how to use Inventor effectively and efficiently.
- More than 190 exercises to reinforce the chapter topics. These exercises also build on previously learned material.
- Chapter tests for review of tools and key Inventor concepts.
- A large selection of modeling and drafting problems to supplement each chapter. Problems are presented as 3D illustrations, industrial drawings, and engineering sketches.

Learning Objectives identify key items you will learn in the chapter.

Command Entry Graphics show ribbon, **Application Menu**, keyboard shortcuts, and other tool entry options.

Illustrations, including Inventor "screen shots" and model illustrations, make learning easy.

Professional Tips increase your productivity in using Inventor tools and techniques.

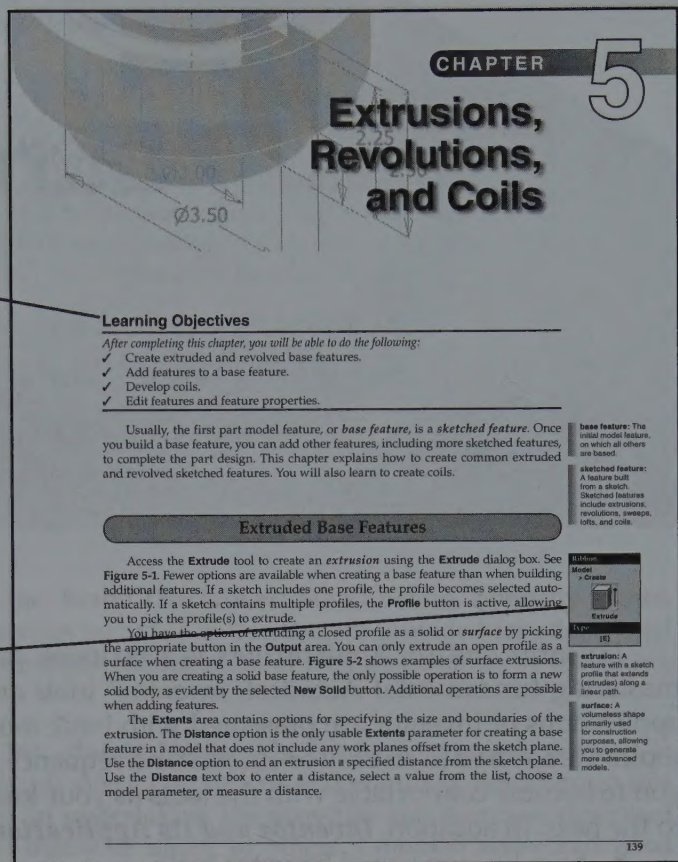**Exercise References** direct you to exercises on the Student Web site.

CHAPTER 5

Extrusions, Revolutions, and Coils

Ø3.50

Learning Objectives

After completing this chapter, you will be able to do the following:
✓ Create extruded and revolved base features.
✓ Add features to a base feature.
✓ Develop coils.
✓ Edit features and feature properties.

Usually, the first part model feature, or *base feature*, is a *sketched feature*. Once you build a base feature, you can add other features, including more sketched features, to complete the part design. This chapter explains how to create common extruded and revolved sketched features. You will also learn to create coils.

base feature: The initial model feature, on which all others are based.

sketched feature: A feature built from a sketch. Sketched features include extrusions, revolutions, sweeps, lofts, and coils.

Extruded Base Features

Access the **Extrude** tool to create an *extrusion* using the **Extrude** dialog box. See **Figure 5-1.** Fewer options are available when creating a base feature than when building additional features. If a sketch includes one profile, the profile becomes selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile(s) to extrude.

You have the option of extruding a closed profile as a solid or *surface* by picking the appropriate button in the **Output** area. You can only extrude an open profile as a surface when creating a base feature. Figure 5-2 shows examples of surface extrusions. When you are creating a solid base feature, the only possible operation is to form a new solid body, as evident by the selected **New Solid** button. Additional operations are possible when adding features.

The **Extents** area contains options for specifying the size and boundaries of the extrusion. The **Distance** option is the only usable **Extents** parameter for creating a base feature in a model that does not include any work planes offset from the sketch plane. Use the **Distance** option to end an extrusion a specified distance from the sketch plane. Use the **Distance** text box to enter a distance, select a value from the list, choose a model parameter, or measure a distance.

Ribbon
Model
> Create

Extrude

Type:
[E]

extrusion: A feature with a sketch profile that extends (extrudes) along a linear path.

surface: A volumeless shape primarily used for construction purposes, allowing you to generate more advanced models.

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Figure 5-3.
Choose an extrusion direction appropriate for the application.

Distance occurs in a positive direction from the sketch plane

Distance occurs in a negative direction from the sketch plane

Divides the distance equally on both sides of the sketch plane
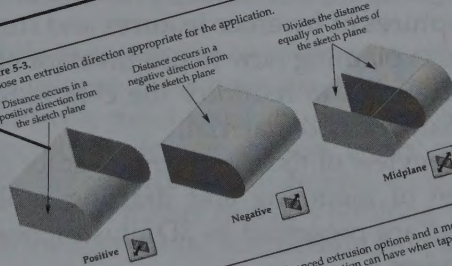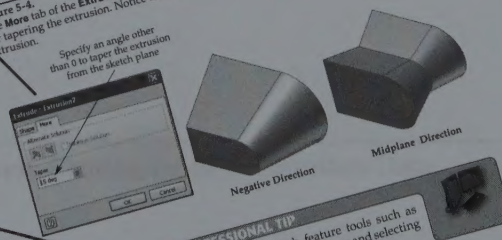
Positive

Negative

Midplane

Figure 5-4.
The **More** tab of the **Extrude** dialog box provides advanced extrusion options and a method for tapering the extrusion. Notice the significant impact direction can have when tapering an extrusion.

Specify an angle other than 0 to taper the extrusion from the sketch plane

Negative Direction

Midplane Direction

PROFESSIONAL TIP

Once you complete a sketch, access sketch feature tools such as **Extrude** from the sketch environment by right-clicking and selecting a tool from the **Create Feature** cascading submenu.

Exercise 5-1
Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-1.

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Chapter 5 Extrusions, Revolutions, and Coils

PROFESSIONAL TIP

The [Esc] key provides an effective way to exit a tool. You may also be able to right-click and select an option such as **Done**, or access another tool to exit a tool. [Delete] allows you to remove an item and may be the only way to delete certain selections or settings from a dialog box. Press [Enter] or the space bar to access the previously used tool.

Supplemental Material Keyboard Shortcuts
For a list of default keyboard shortcuts, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Keyboard Shortcuts**.
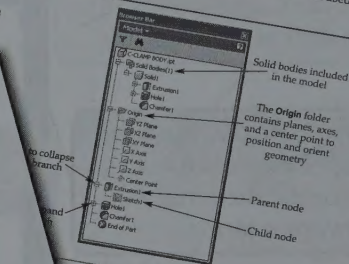
Browser

browser (browser bar): A panel that displays all the items in the current model or drawing.

The *browser*, or *browser bar*, provides a historical reference of file content. For example, the part file browser displays all elements of the part model in the order in which you create the items. See **Figure 1-30**. The number and type of items available in the browser vary depending on the current file, work environment, and design stage. You can make changes to file content directly in the graphics window or from the browser.

parent node: An item in the tree structure, similar to a folder, that is associated with subordinate child nodes.

child node: Subordinate nodes that create, are associated with, or are consumed by a parent node item.

NOTE

Items in the browser are listed in the order they were created or inserted, although it is possible and sometimes necessary to drag and drop browser items up or down in the list to change the order.

A tree structure, with *parent nodes* and *child nodes*, arranges many of the items in the browser. For example, a sketched feature is a parent node, and the sketch used to

Figure 1-30.
The browser contains information regarding the content of a model or drawing. This browser is associated [...]

Solid bodies included in the model

The **Origin** folder contains planes, axes, and a center point to position and orient geometry

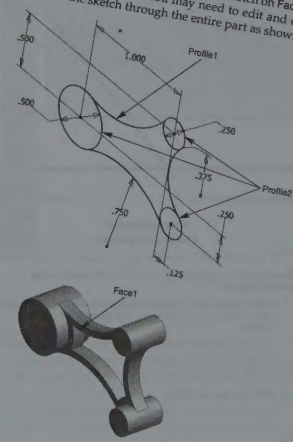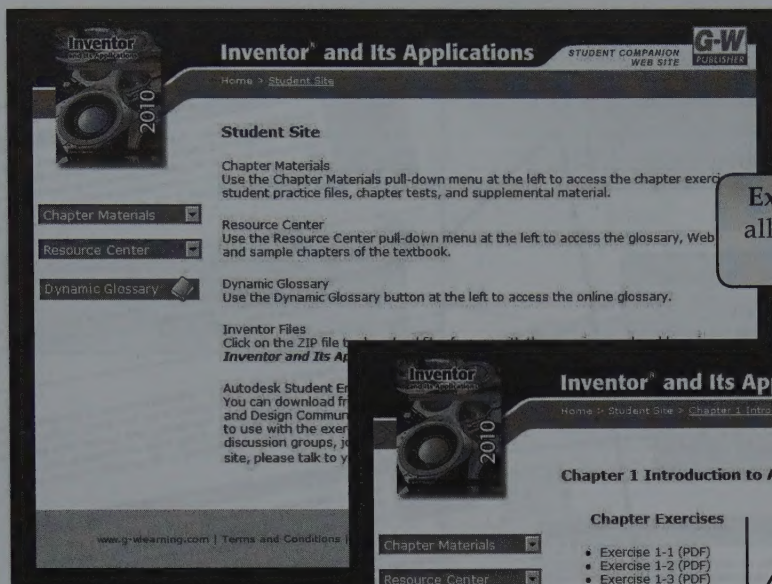Parent node

Child node

to collapse branch

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Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. What is a profile?
2. Describe the purpose of a path.
3. What are the two basic steps in sketch development?
4. Why are sizes often approximate in initial sketch geometry?
5. What is the difference between a closed loop and an open loop?
6. What is a 2D plane?
7. Define coordinate system.
8. Where is the center point of the model coordinate system?
9. Identify the plane on which a sketch forms by default when you sta[...]
   model and explain its rotation.
10. Name the tool used when you finish creating a sketch and are re[...]
    feature work environment.
11. Define the terms zooming out and zooming in.
12. Explain how to zoom out and in using the mouse wheel.
13. Explain what panning is and how to use the mouse to pan.
14. Briefly describe how to project the center point in the Origin[...]
    sketch plane.
15. What are geometric constraints?
16. Give at least two common uses of geometric construction[...]
17. What is an inferred geometric constraint?
18. Which geometric constraint specifies that objects s[...]
    never intersect, no matter how long they become?
19. Which geometric constraint defines a 90° angle be[...]
    ellipse axes?
20. What is a coincident constraint? Give an examp[...]
21. Briefly describe how to create a line with a tan[...]
22. What is a spline?
23. Which tool should you use to create a circl[...]
    one point only?
24. What three items do you need to specif[...]
    tool?
25. What is a regular polygon?
26. Explain the difference between roun[...]
27. Briefly describe how to add an equal[...]
28. What is the difference between ske[...]
29. Name the tool used to add a basic[...]
30. Briefly describe how to align ske[...]

Chapter 3 Introduction

▼ Intermediate
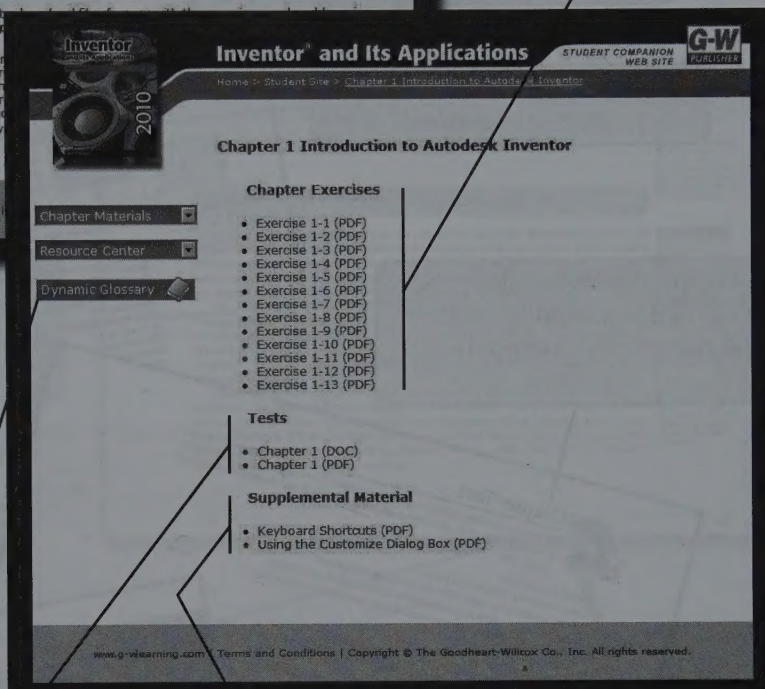
4. File: P4-4.ipt
   Save as: P5-4.ipt
   Title: SUPPORT BRACKET
   Specific instructions: Extrude Profile1 .125" midplane. Share Sketch1. Extrude the three circle profiles (Profile2) .375" midplane. Open a sketch on Face1 and offset the automatically projected edges .0625". You may need to edit and constrain sketch geometry. Cut-extrude the sketch through the entire part as shown.
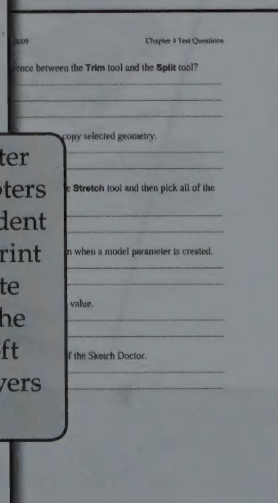
.500
1.000
.500
.250
.375
Profile1
Profile2
.750
.250
.125

Face1

Problems — Chapter 5

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Features of the Textbook

Student Site

Chapter Materials
Use the Chapter Materials pull-down menu at the left to access the chapter exercises, student practice files, chapter tests, and supplemental material.

Resource Center
Use the Resource Center pull-down menu at the left to access the glossary, Web links, and sample chapters of the textbook.

Dynamic Glossary
Use the Dynamic Glossary button at the left to access the online glossary.

Inventor Files
Click on the ZIP file to download the files for use with the exercises and problems in *Inventor and Its Applications*.

Autodesk Student Engineering
You can download free software from the Autodesk Student Engineering and Design Community to use with the exercises, join discussion groups, join this site, please talk to your...

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Inventor® and Its Applications — STUDENT COMPANION WEB SITE — G-W PUBLISHER

Home > Student Site > Chapter 1 Introduction to Autodesk Inventor

Chapter 1 Introduction to Autodesk Inventor

Chapter Exercises

- Exercise 1-1 (PDF)
- Exercise 1-2 (PDF)
- Exercise 1-3 (PDF)
- Exercise 1-4 (PDF)
- Exercise 1-5 (PDF)
- Exercise 1-6 (PDF)
- Exercise 1-7 (PDF)
- Exercise 1-8 (PDF)
- Exercise 1-9 (PDF)
- Exercise 1-10 (PDF)
- Exercise 1-11 (PDF)
- Exercise 1-12 (PDF)
- Exercise 1-13 (PDF)

Tests

- Chapter 1 (DOC)
- Chapter 1 (PDF)

Supplemental Material

- Keyboard Shortcuts (PDF)
- Using the Customize Dialog Box (PDF)

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Exercises. Chapter exercises allow you to practice chapter skills and techniques.
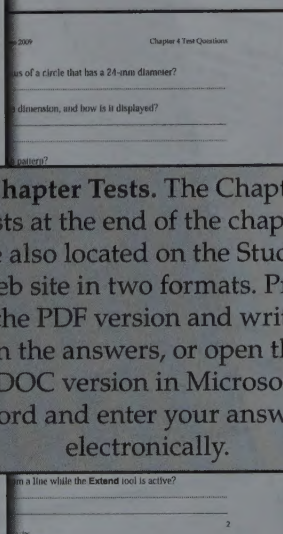
Dynamic Glossary. Find the meaning of terms fast using this electronic dictionary of Inventor terms.
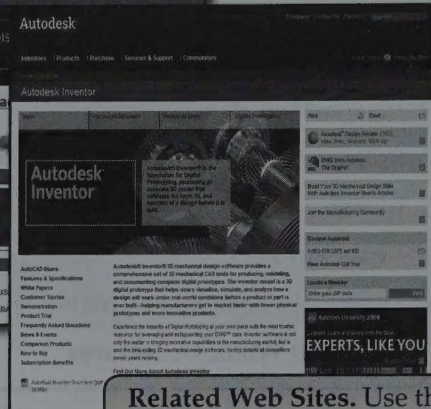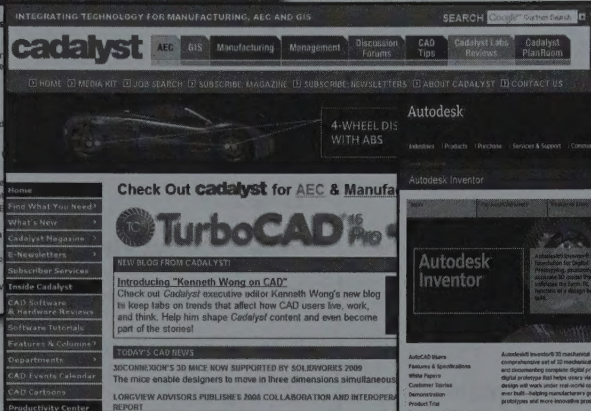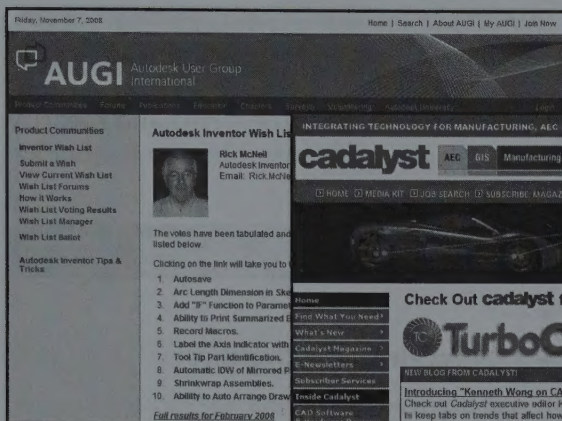
Supplemental Material. Organized by chapter, these supplements provide additional information about topics discussed in the textbook.
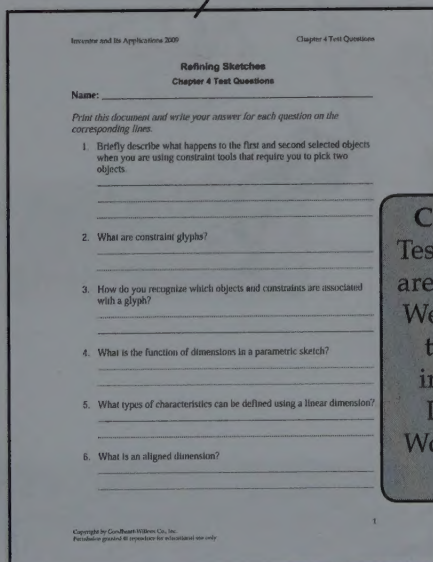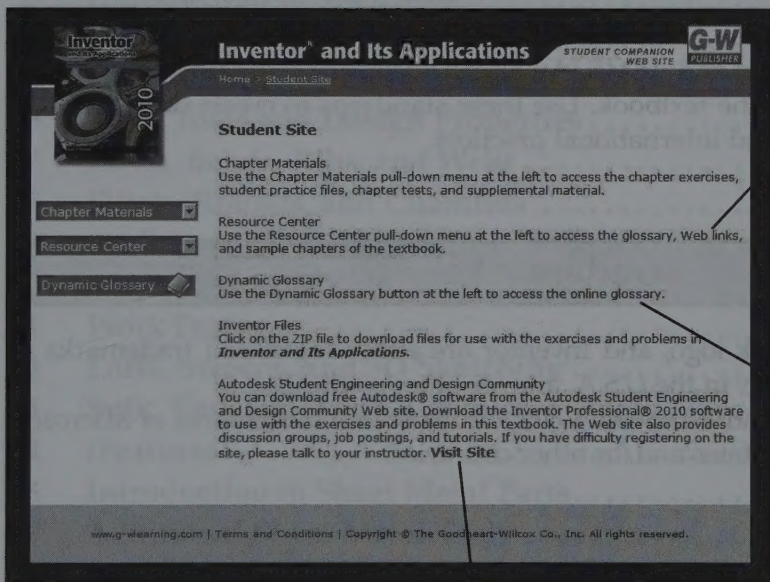
Chapter Tests. The Chapter Tests at the end of the chapters are also located on the Student Web site in two formats. Print the PDF version and write in the answers, or open the DOC version in Microsoft Word and enter your answers electronically.

Inventor and Its Applications 2009 — Chapter 4 Test Questions

Refining Sketches
Chapter 4 Test Questions

Name: _____

Print this document and write your answer for each question on the corresponding lines.

1. Briefly describe what happens to the first and second selected objects when you are using constraint tools that require you to pick two objects.

2. What are constraint glyphs?

3. How do you recognize which objects and constraints are associated with a glyph?

4. What is the function of dimensions in a parametric sketch?

5. What types of characteristics can be defined using a linear dimension?

6. What is an aligned dimension?

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Related Web Sites. Use these links to access a variety of CAD/drafting Web sites.

Glossary. Print this traditional glossary and use it for reference as you work through the chapters.

Inventor Software. Pick this button to access a Web site from which you can download the Inventor 2010 Professional software at no cost for use with this book.

Fonts Used in This Textbook

Different typefaces are used throughout this textbook to define terms and identify AutoCAD commands. The following typeface conventions are used in this textbook:

Text Element	Example
Inventor tools	**LINE** tool
Buttons on the Inventor ribbon tabs	**View** > **Object Visibility** > **Sketches**
Inventor dialog boxes	**Style and Standard Editor** dialog box
Keyboard keys	[Ctrl]+[1] key combination
File names, folders, and paths	C:\Program Files\AutoCAD 2009\mydrawing.dwg
Microsoft Windows features	Start menu, Programs folder
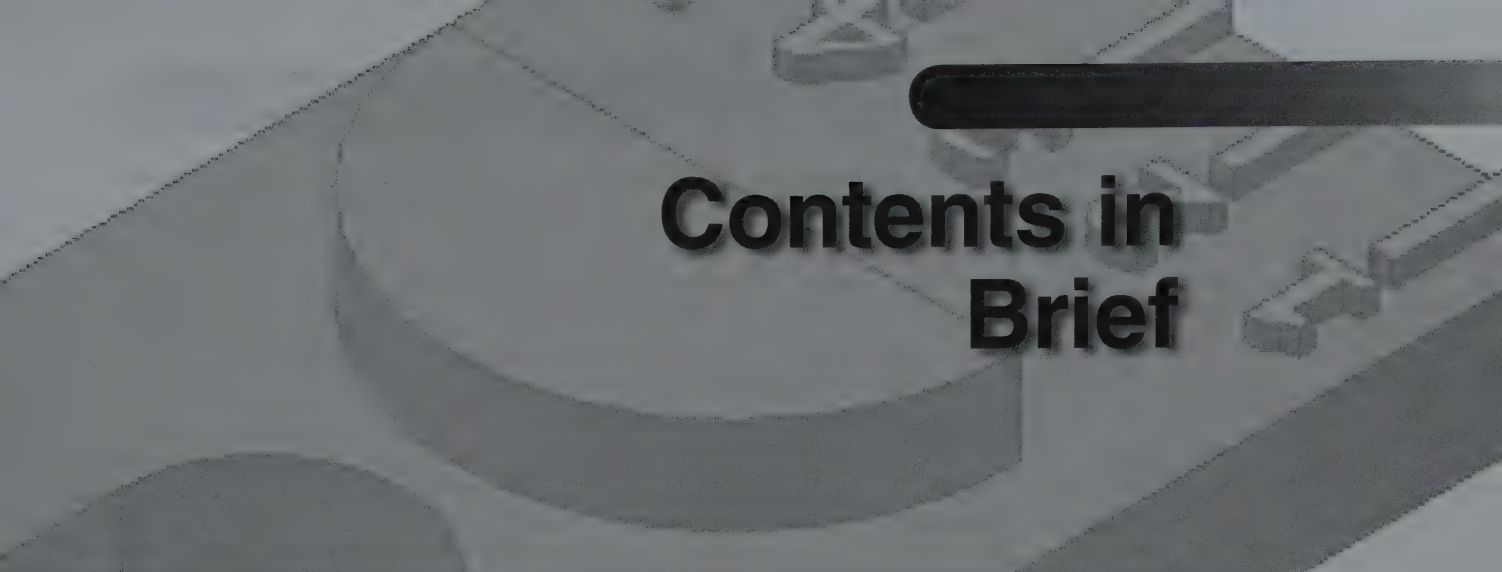
Other Text References

For additional information, standards from organizations such as ANSI (American National Standards Institute) and ASME (American Society of Mechanical Engineers) are referenced throughout the textbook. Use these standards to create drawings that follow industry, national, and international practices.

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Contents in Brief

About the Author

David P. Madsen is vice president of Madsen Designs Inc. (www.madsendesigns.com), and operates the Madsen Designs Inc. consulting service. Dave has been a professional design drafter since 1996 and has extensive experience in a variety of drafting, design, and engineering disciplines. Dave has provided drafting and computer-aided design and drafting instruction to secondary and postsecondary learners since 1999 and has considerable curriculum and program coordination and development experience. Dave holds a Master of Science degree in Educational Policy, Foundations, and Administrative Studies with a specialization in Postsecondary, Adult, and Continuing Education; a Bachelor of Science degree in Technology Education; and an Associate of Science degree in General Studies and Drafting Technology. Dave is the author of *Inventor and Its Applications* and coauthor of Goodheart-Willcox's *Architectural Drafting Using AutoCAD, AutoCAD and Its Applications: Basics* and *Comprehensive,* and *Geometric Dimensioning and Tolerancing.*

Acknowledgments

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Related Web Links

Introduction to Autodesk Inventor

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Describe the primary Autodesk Inventor file types.
✓ Explain the concept of parametric design and drafting.
✓ Identify the elements of a part model.
✓ Begin new files and open, save, and close files.
✓ Describe the Autodesk Inventor user interface.
✓ Manage multiple open documents.
✓ Access application options and locate help resources.

Autodesk Inventor, referred to as *Inventor* in this textbook, is a powerful *computer-aided design and drafting (CADD)* system that provides three-dimensional (3D) *solid modeling* and two-dimensional (2D) *drawing* capabilities. See **Figure 1-1**. Inventor allows you to start with basic product design ideas and end with a virtual prototype and a complete set of working drawings. You can adapt Inventor to conform to a variety of mechanical design and drafting *standards*. This textbook focuses on American National Standards Institute (ANSI) and the American Society of Mechanical Engineers (ASME) design and drafting standards. This textbook also presents other common standards, such as the American Welding Society (AWS) standards, when appropriate.

Inventor Files

Inventor offers unique file formats to prepare 3D part, assembly, and presentation models, and 2D drawings. Each file type provides *tools* and *options* related to specific design and drafting requirements. A basic project may require a part file or part and drawing files. Complex projects may require hundreds of part, assembly, presentation, and drawing files.

computer-aided design and drafting (CADD): The process of using a computer with software to design and produce models and drawings.

solid model: A model that contains object volume and mass data used to analyze internal and external object characteristics.

drawing: 2D representation of a model containing views, dimensions, and annotations.

standards: Guidelines containing operating procedures, drawing techniques, and record keeping methods.

tool (command): An instruction issued to the computer to complete a specific task. For example, the **LINE** tool is used to draw lines.

option: A choice associated with a tool, or an alternative function of a tool.

3D Solid Model

2D Drawing

Parts

A part file allows you to create a *part* model. See **Figure 1-2**. Assembly files reference parts for assembly models, and drawing files reference parts for part drawings. Specialized sheet metal tools are available for preparing sheet metal part models. Part files use the .ipt file extension.

Assemblies

An assembly file allows you to reference *component* files to create an assembly model. See **Figure 1-3**. Specialized *weldment* tools are available for preparing weldment models. You can develop separate part and assembly files for insertion into an

Figure 1-2.
A C-clamp body
part model.
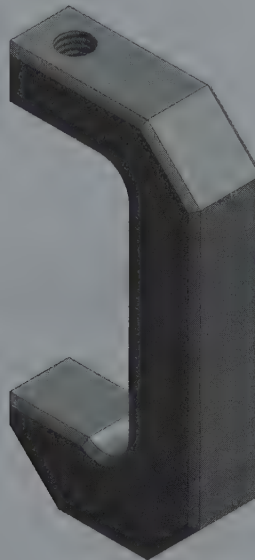
Figure 1-3.
A C-clamp assembly
model.
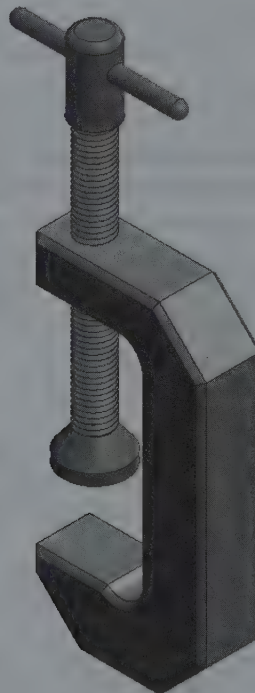
assembly. This is an example of a process that some designers refer to as ***bottom-up design***. An alternative is to create and save a part and assembly files in place within an assembly file. Some designers refer to this process as ***top-down design***. Assembly files reference assemblies as subassemblies for assembly models, presentation files reference assemblies for exploded and animated assembly models, and drawing files reference assemblies for assembly drawings. Assembly files use the .iam file extension.

bottom-up design: A design approach that brings individual components together to form an assembly.

top-down design: A design approach in which the assembly controls, or produces, individual components.

Presentations

A presentation file allows you to reference an assembly file to create an exploded, animated, and stylized assembly model. See **Figure 1-4**. Presentations show how parts and subassemblies interact within the assembly. Drawing files reference presentations for exploded assembly drawings. Presentation files use the .ipn extension.

Figure 1-4.
An example of a
C-clamp presentation
model that contains
trails, or connection
graphics, between
components to show
relative position and
movement in the
assembly.

Figure 1-5.
An example of a drawing that documents the design of a C-clamp body. A drawing should
provide enough information to describe the product without confusion.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
3. ALL CHAMFERS 1.6 X 45°.
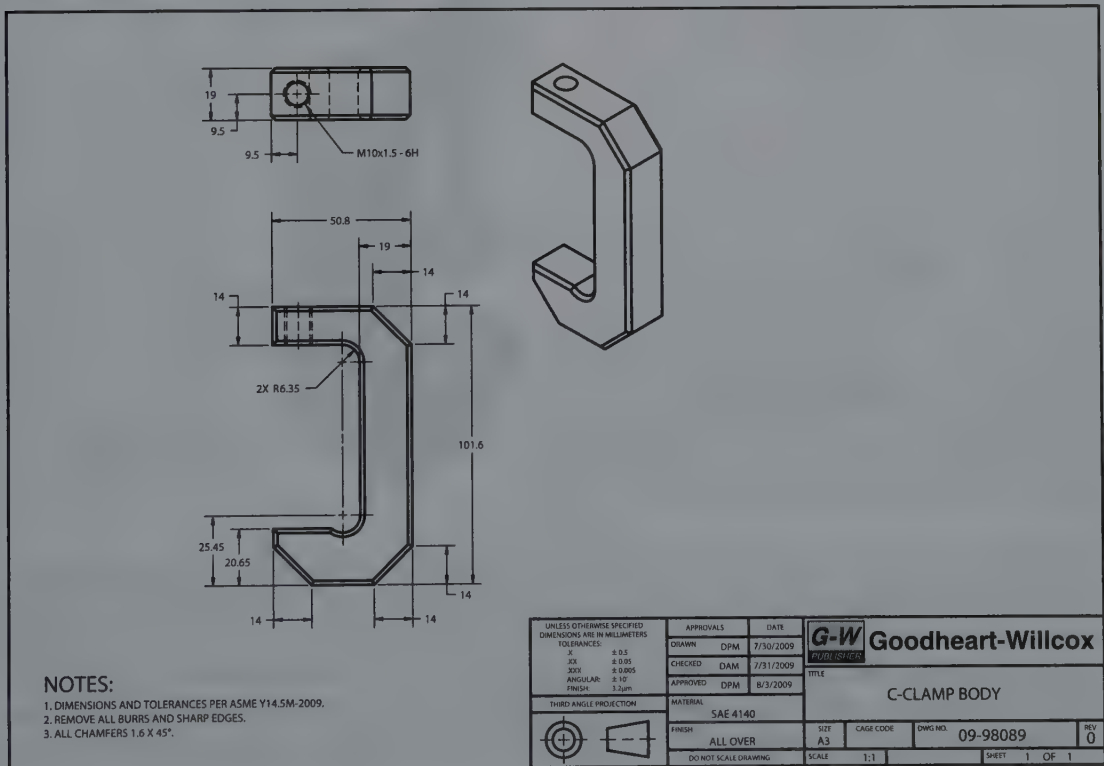
Drawings

A drawing file allows you to prepare a 2D drawing. See **Figure 1-5**. Drawings
reference existing part, assembly, and presentation files. Drawing files use the .idw or
.dwg extension.

Exercise 1-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-1.

Parametric Fundamentals

Inventor provides *parametric design and drafting* tools for assigning *parameters*, or *constraints*, to objects. The parametric concept, also known as *intelligence*, provides a way to associate objects and limit design changes. You cannot change a parameter that conflicts with other parametric geometry. A database stores and allows you to manage all parameters. Parametric design and drafting allows you to control every aspect of a project during and after the design and documentation processes.

Types of Constraints

Parameters include *geometric constraints* and *dimensional constraints*. You must add constraints to make an object parametric. Well-defined constraints increase design and revision efficiency, place limits on geometry to preserve design intent, and help form geometric constructions. For example, if two holes through a part, drawn as circles, must always be the same size, use a geometric constraint to make the circles equal and a dimensional constraint to size one of the circles. Dimensional constraints create parameters that direct object size and location. The size of both circles changes when you modify the dimensional constraint value. See **Figure 1-6**.

Figure 1-6.
An example of a basic parametric relationship. The dimensional constraint controls the size of both circles with the aid of an equal geometric constraint.
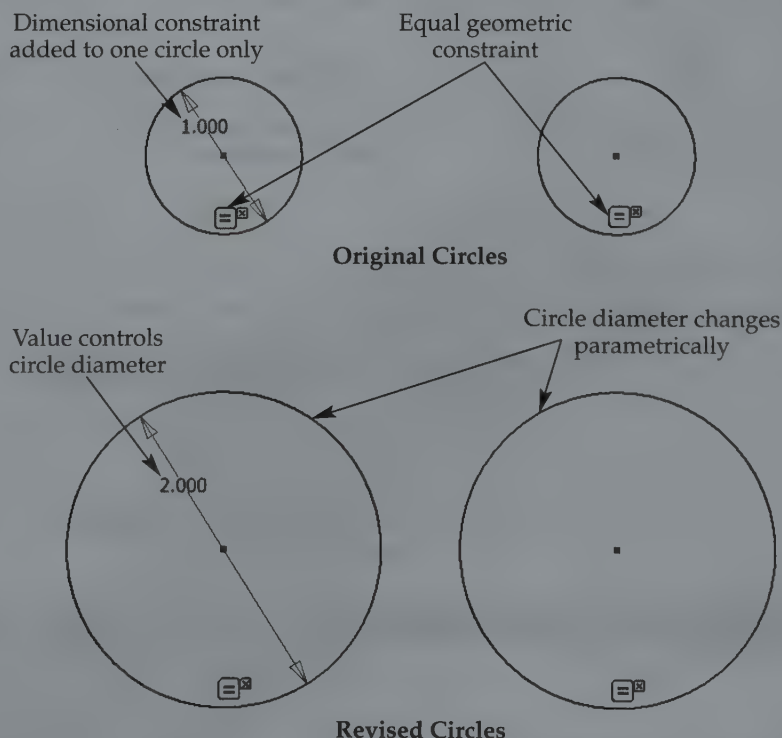
Dimensional constraint added to one circle only

Equal geometric constraint

1.000

Original Circles

Value controls circle diameter

Circle diameter changes parametrically

2.000

Revised Circles

Constraint Levels

Figure 1-7 shows an example of constraint levels, including *under-constrained*, *fully constrained*, and *over-constrained* sketches. You often fully or near fully constrain a model to ensure accuracy. However, if you attempt to over-constrain a model, a message appears allowing you to account for the situation. See **Figure 1-8**. Inventor does not allow you to over-constrain a model, as shown by the *reference dimension* in **Figure 1-7**.

Figure 1-9 shows an extreme example of constraining, primarily for reference. Study the figure to help understand how constraints work, and how applying constraints differs from and complements traditional drafting practices. Typically, you should prepare initial objects as accurately as possible using the many geometric construction tools available, and then add constraints as needed.

PROFESSIONAL TIP

As you learn to use Inventor, you will recognize and work with many parametric relationships. Successful parametric design and drafting requires detailed planning and an understanding of how model and drawing elements relate to each other throughout the design and documentation processes.

Figure 1-7.
Levels of parametric constraints.
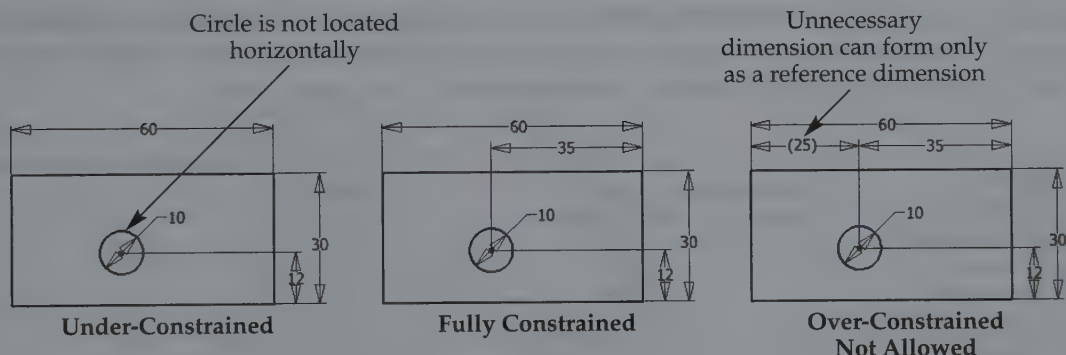
Figure 1-8.
Alert or error messages appear when you attempt to over-constrain or improperly constrain objects.
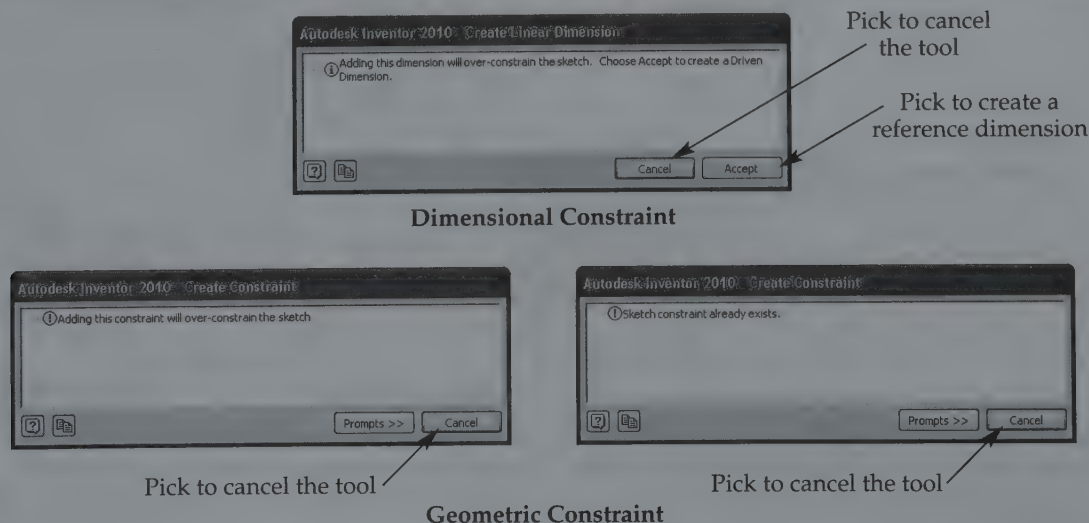
Part Model Elements

Part models allow you to design parts, build assembly models, and prepare part drawings. A part model begins as a sketch or group of sketches used to construct a feature. Add features as necessary to create the final part model. Primary part model features include sketched, placed, work, catalog, and patterned features. You can develop additional model elements, such as surfaces, as needed to build a part model.

Sketches and Sketched Features

Every part model usually contains at least one *sketch* and at least one *sketched feature*. **Figure 1-9** shows an example of a sketch. Normally, the *base feature* is a sketched feature. **Figure 1-10** shows an example of sketched extrusions, a rib, and an embossment added to an extruded base feature.

sketch: A 2D drawing that provides the profile or guide for developing a sketched feature.

sketched features: Features that are built from sketches, such as extrusions, revolutions, sweeps, lofts, and coils.

base feature: The initial model feature, on which all others are based.

Figure 1-9.
An extreme example of constraining a drawing to help you understand how constraints are applied.

Coincident constraints "connect" endpoints

Perpendicular

Parallel

Horizontal

Unconstrained

Geometric Constraints Added

Dimensional Constraints Added
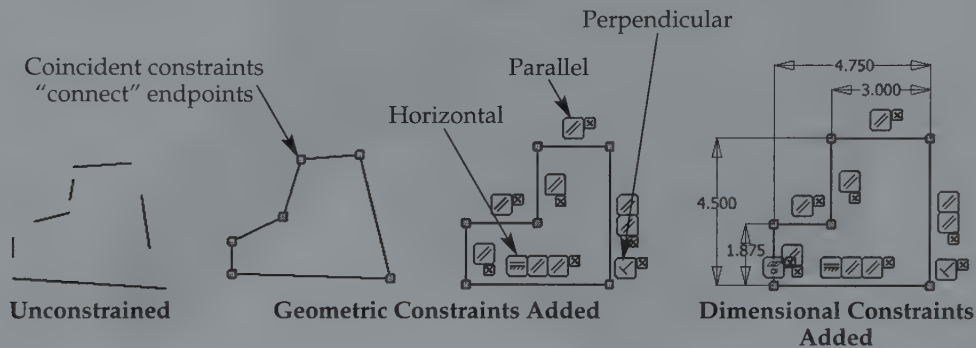
Figure 1-10.
Sketched features added to a sketched base feature.
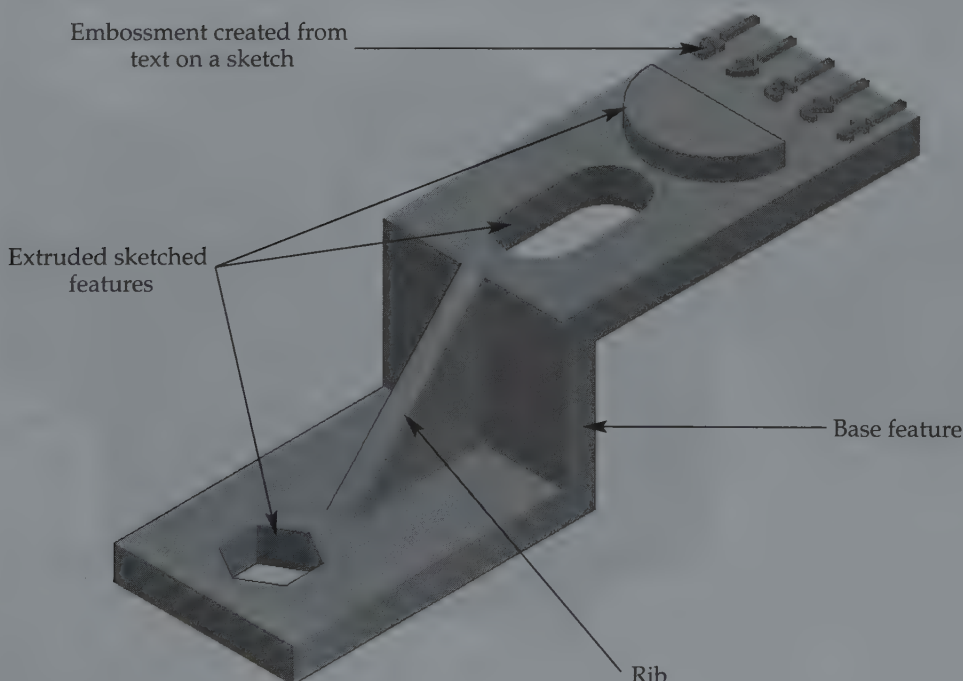
Embossment created from text on a sketch

Extruded sketched features

Base feature

Rib

Placed Features

Adding *placed features* requires specifying size dimensions and characteristics and selecting a location, such as a point or edge. No sketch is necessary. **Figure 1-11** shows examples of common placed features, including a chamfer, a fillet, and threads.

Work Features

Add *work features* to existing features or in 3D space for construction and reference purposes. Work features are often necessary to build a model when needed geometry is unavailable or to form specific parametric relationships. **Figure 1-12** shows basic work features, including a plane, axis, and point.

Figure 1-11. Chamfers, fillets, and threads are a few of the available placed features.
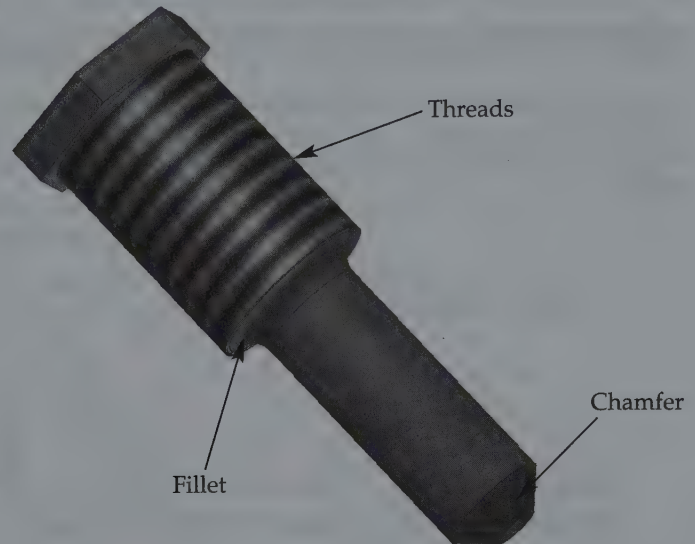
Figure 1-12. Work features allow you to build and control other features.
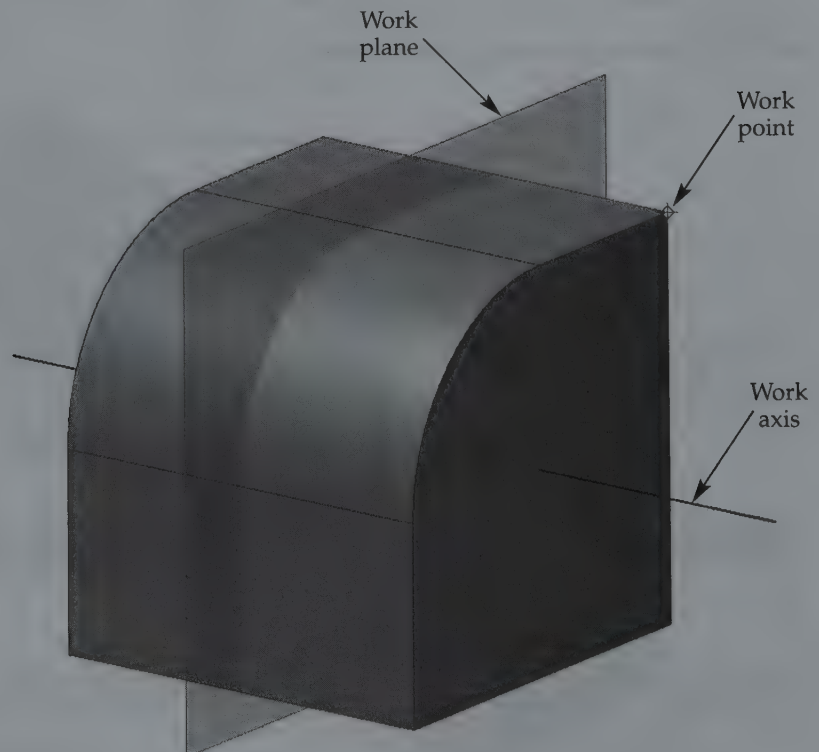
Catalog Features

Catalog features are similar to placed features, but they are often more complex and represent a specific element, such as a certain boss, slot, or stock item. Catalog features include design elements such as *iFeatures* and *derived components*. **Figure 1-13** shows an example of an iFeature.

Feature Patterns

Once you create a feature, you have the option of creating a *feature pattern*. Patterning features often saves considerable time while forming and maintaining parametric relationships. Feature patterns include rectangular and circular patterns and mirrored features. See **Figure 1-14**.

Figure 1-13.
An example of a catalog feature that includes three holes and a round.

Catalog feature
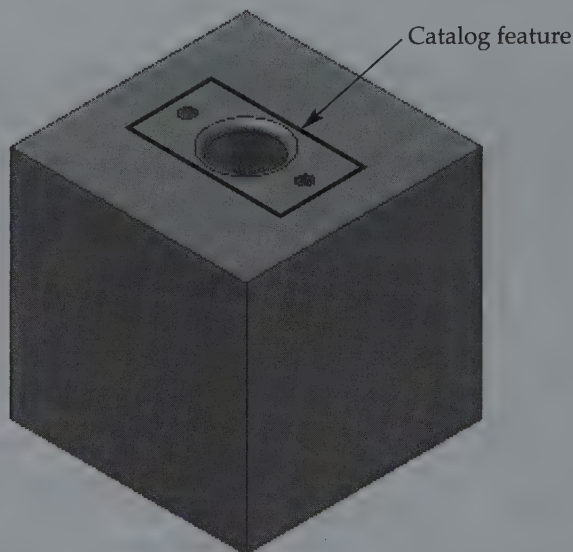
Figure 1-14.
Use patterns to create multiple copies of a feature or group of features.
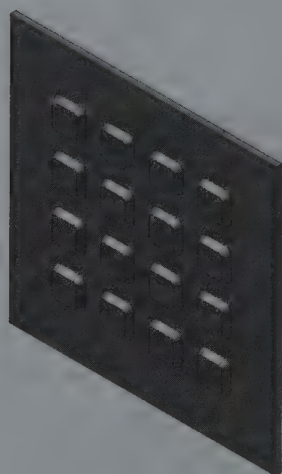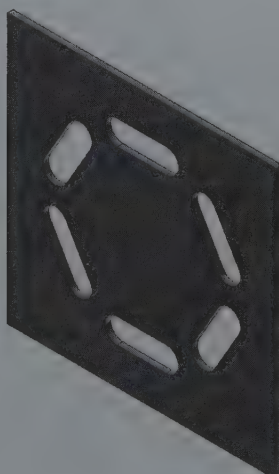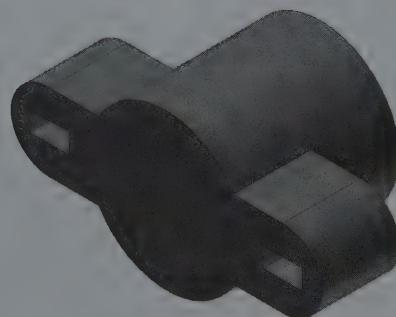
Rectangular Pattern   Circular Pattern   Mirrored Feature

Exercise 1-2

Getting Started

double-click: Quickly press the left mouse button twice to select.

icon: A small graphic representing an application, file, or tool.

pick (click): Use the left mouse button to select.

button: A "hot spot" on the screen that can be picked to access an application, tool, or option.

hover: Pause the cursor over an item to display information or options.

default: A value maintained by the computer until changed.

toolbar: An interface item that contains tool buttons or drop-down lists.

ribbon: The palette that extends across the top of the Inventor interface and contains multiple tabs for convenient tool access.

To start Inventor, *double-click* on the Autodesk Inventor Professional 2010 desktop *icon* that appears during installation. You can also *pick* the Start *button* in the lower-left corner of the Windows desktop. Then *hover* over or pick Programs, select Autodesk, select Autodesk Inventor 2010, and finally select Autodesk Inventor Professional 2010.

> **NOTE**
>
> Due to installation, operating system, and display variables, you may see illustrations in this textbook that appear slightly different from those on your screen.

Figure 1-15 shows the *default* application window that appears when you launch Inventor. The **Application Menu**, **Quick Access** *toolbar*, and *ribbon* provide options for beginning a new file and opening an existing file. The default application window provides additional options, including those for saving and closing files, controlling system settings, and exiting Inventor.

Figure 1-15.
The default application window that appears when you launch Inventor.
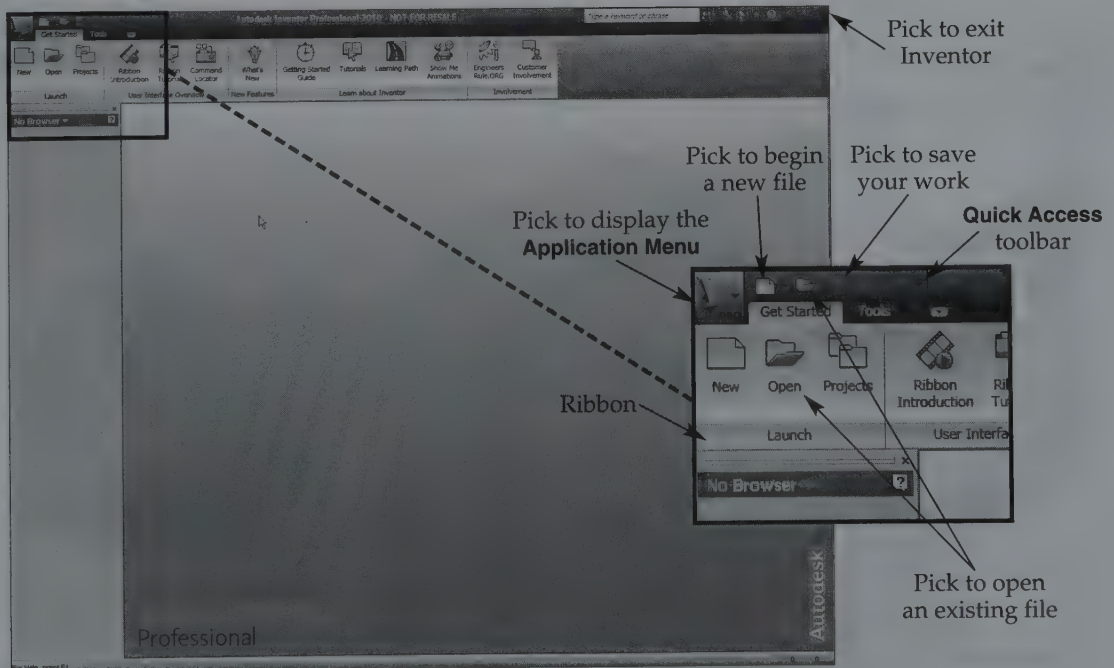
Figure 1-16.
Use the **New File**
dialog box to access
templates for
creating new models
and drawings.

Contains templates based on choosing English
or metric during program installation

Contains templates based on U.S. Customary units

Contains templates based on metric units

Standard
drawing

Standard
drawing

Standard
part

Sheet metal
part

Standard
assembly

Standard
presentation

Weldment
(welded
assembly)

Pick to open an existing file
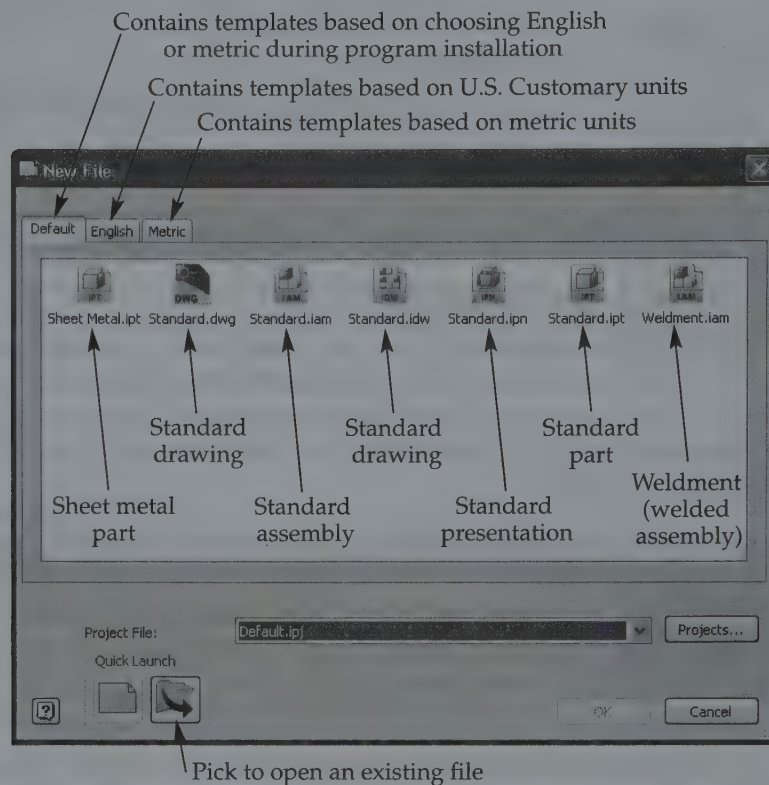
Beginning a New File

To begin a new file, pick the **New** button from the **Application Menu, Quick Access** toolbar, or **Launch** panel of the **Get Started** *ribbon tab*. An alternative is to type [Ctrl]+[N]. The **New File** dialog box shown in **Figure 1-16** appears, allowing you to reference a *template* to create a new part, assembly, presentation, or drawing.

> **NOTE**
>
> The **New** *flyout* on the **Quick Access** toolbar, shown in **Figure 1-15**, provides access to **Assembly, Drawing, Part**, and **Presentation** templates named Standard. The **New** menu in the **Application Menu** provides the same options. Chapter 2 explains the content and location of these templates.

By default, the **New File** dialog box includes **Default, English**, and **Metric** tabs. Select a tab to display templates for beginning new files. The **English** and **Metric** tabs include templates with units and settings associated with specific design and drafting standards. When you install Inventor, you can direct the **Default** tab to display the same templates provided in the **English** or **Metric** tab.

A sheet metal template begins a new part file preset for sheet metal modeling. A weldment template begins a new assembly file preset for weldment modeling. An .idw or .dwg template begins a new drawing. The .dwg format is for direct use in AutoCAD. To begin a new file, double-click on the template icon in the *list box* or pick the icon and select the **OK** button.

> **NOTE**
>
> The tabs and templates displayed in the **New File** dialog box are adjustable, as described in Chapter 2.

ribbon tab: A collection of tool panels on the ribbon accessed by picking the collection's name near the top of the ribbon.

templates: Files with predefined settings used to begin new documents.

flyout: Set of related buttons that appears when you pick the arrow next to certain tool buttons.

list box: A boxed area that contains a list of items or options from which to select.

Exercise 1-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-3.

Opening an Existing File

To open an existing file, pick the **Open** button from the **Application Menu**, **Quick Access** toolbar, or **Launch** panel of the **Get Started** ribbon tab. An alternative is to type [Ctrl]+[O]. The **Open** dialog box shown in **Figure 1-17** appears, allowing you to locate and open existing parts, assemblies, presentations, and drawings, as well as other files supported by Inventor.

> **NOTE**
>
> You can open existing files directly from Microsoft Windows Explorer. The **Application Menu** provides a recent documents function for opening recently used files.

As explained further in Chapter 2, the active Inventor project controls the folder that initially appears when you access the **Open** dialog box. However, you can navigate to and open any file on your computer or the network. Use the **Files of type** *drop-down list* to filter the file types to display. Then use the **Look in** drop-down list, **Go To Last Folder Visited**, **Up One Level**, **Create New Folder**, and **View Menu** buttons and the **Preview** area to locate a file to open. Once you select a file, the **Options...** button may become enabled. Pick the button to display the **File Open Options** dialog box, where you can adjust opening options specific to the selected file type. To open a file, double-click on the file icon or pick the icon and then pick the **Open** button. You can select multiple files to open at the same time.

drop-down list: A list of options that appears when you pick a button that contains a down arrow.

Figure 1-17.
Use the **Open** dialog box to open existing Inventor and supported files. Inventor allows you to open or import a variety of CADD file formats.
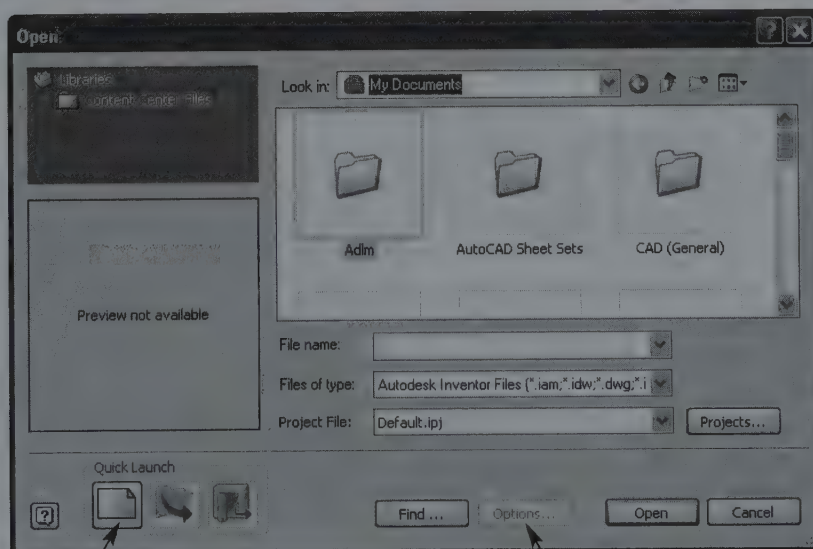
Pick to begin a new file

This button becomes available after certain files are selected

To search for a file, pick the **Find...** button in the **Open** dialog box to open the **Find: Inventor Files** dialog box. You can also use the Windows search tool to find a file.
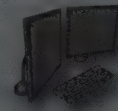
Double-click on an Inventor file in Microsoft Windows Explorer to launch Inventor, if it is not already running, and open the selected file.

Exercise 1-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-4.

Saving Your Work

You must save a file to access the file for future use. Develop the practice of saving each file in an appropriate location using a descriptive name or code file as the first step in creating a model or drawing. Then, as you work, save the file at least every five to ten minutes to avoid losing work in case of a sudden loss of power or a computer problem.

Using the Save Tool

To use the **Save** tool, pick the **Save** button from the **Application Menu** or **Quick Access** toolbar. An alternative is to type [Ctrl]+[S]. The **Save** tool opens the **Save As** dialog box, shown in **Figure 1-18**, if a file is unsaved, or resaves a saved file without displaying the **Save As** dialog box. To save a file using the **Save As** dialog box, use the **Look in** drop-down list, and **Go To Last Folder Visited**, **Up One Level**, **Create New Folder**, and **View Menu** buttons to locate the file or adjust the folder.

Figure 1-18.
The **Save As** dialog box appears if the current file is unsaved.
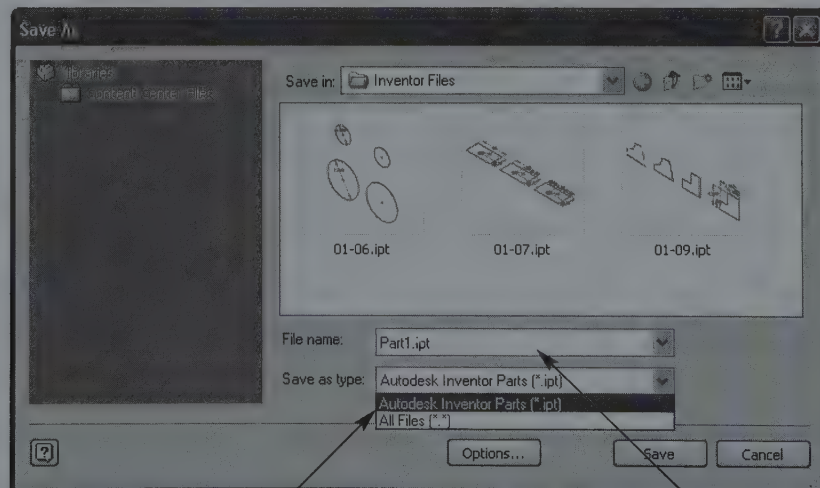
The list of file types is restricted to match the current type

Type the new file name in this box

Type a name for the file in the **File name** *text box*. The file name is a file property that is referenced throughout design and documentation. Set up a system that allows you to determine the content of a file by the file name. File names often identify a product by name and number, such as VICE-101 or 6DT1009. Use a standard file naming system that contains a clear and concise reference to the project, part number, process, sheet number, and revision level, depending on the product and standards.

The file type listed in the **Files of type** drop-down list should display the current file type. Pick the **Options...** button to open the **File Save Options** dialog box. For Inventor files, the **File Save Options** dialog box controls the method of generating the preview image that displays in the **Preview** area of the **File Open** dialog box. Pick the **Save** button to complete the save.

> **NOTE**
>
> File names are not case sensitive and can include a maximum of 256 characters, most alphabetic and numeric characters, spaces, and most punctuation symbols. You do not have to include the file extension with the file name. File names cannot include quotation mark ("), asterisk (*), question mark (?), forward slash (/), and backward slash (\) characters.

Using the Save As Tool

To save a saved file using a different name, access the **Save As** tool by picking the **Save As** button from the **Application Menu**. The **Save As** dialog box appears, allowing you to rename the file. The original file closes and the new file opens. For example, to use a part named LARGE PLATE as a basis for a small plate design, use the **Save As** tool to save the LARGE PLATE file as SMALL PLATE. The LARGE PLATE file closes and the SMALL PLATE file appears, ready for modification.

> **NOTE**
>
> You will use the **Save As** tool throughout the exercises and problems in this textbook to save an existing file using a different name.

Using the Save Copy As Tool

To save a copy of a saved file using a different name without closing the original file, access the **Save Copy As** tool by picking the **Save Copy As** button from the **Save As** menu of the **Application Menu**. The **Save Copy As** dialog box appears, allowing you to create a copy of the open file, usually with a different name. The original file remains open. The copy is saved to the specified location and does not open.

Use the **Save Copy As** tool to make a copy as a design state backup or for later use while you continue working with the original file. The **Save Copy As** tool also allows you to save a file in an alternative format, such as IGES, JPEG, PDF, or STEP. Saving a copy in a non-Inventor format is often necessary if you are collaborating with others, or in order to create an image from the on-screen display. Use the **Save as type** drop-down list to select the appropriate file type.

Closing a File

Close a file to finish working with the file without exiting Inventor. To close the active file, pick the **Close** button on the right side of the file window title bar. An alternative is to pick the **Close** button from the **Application Menu**.

Exiting Inventor

To exit Inventor, pick the **Close** button on the right side of the application window title bar, double-click the **Application Menu** button, or select the **Exit Inventor** button from the **Application Menu**. When you close a file or exit Inventor, *alert* boxes may appear, depending on file status and current work environment. For example, if a file remains unsaved, an alert message asks if you want to save the file before closing.

Exercise 1-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-5.

The Inventor Interface

Interface, or *user interface*, items include devices to input data, such as the keyboard and mouse, and devices to receive outputs, such as the monitor. Inventor uses a Windows-style *graphical user interface (GUI)* with an **Application Menu**, **Quick Access** toolbar, ribbon, dialog boxes, and Inventor-specific items such as the *graphics window*. See **Figure 1-19**. Inventor includes other specialized interface items commonly used throughout the design and documentation process, including the navigation bar and view cube. You will learn to use these tools when applicable.

Figure 1-19.
Elements of the Inventor interface.
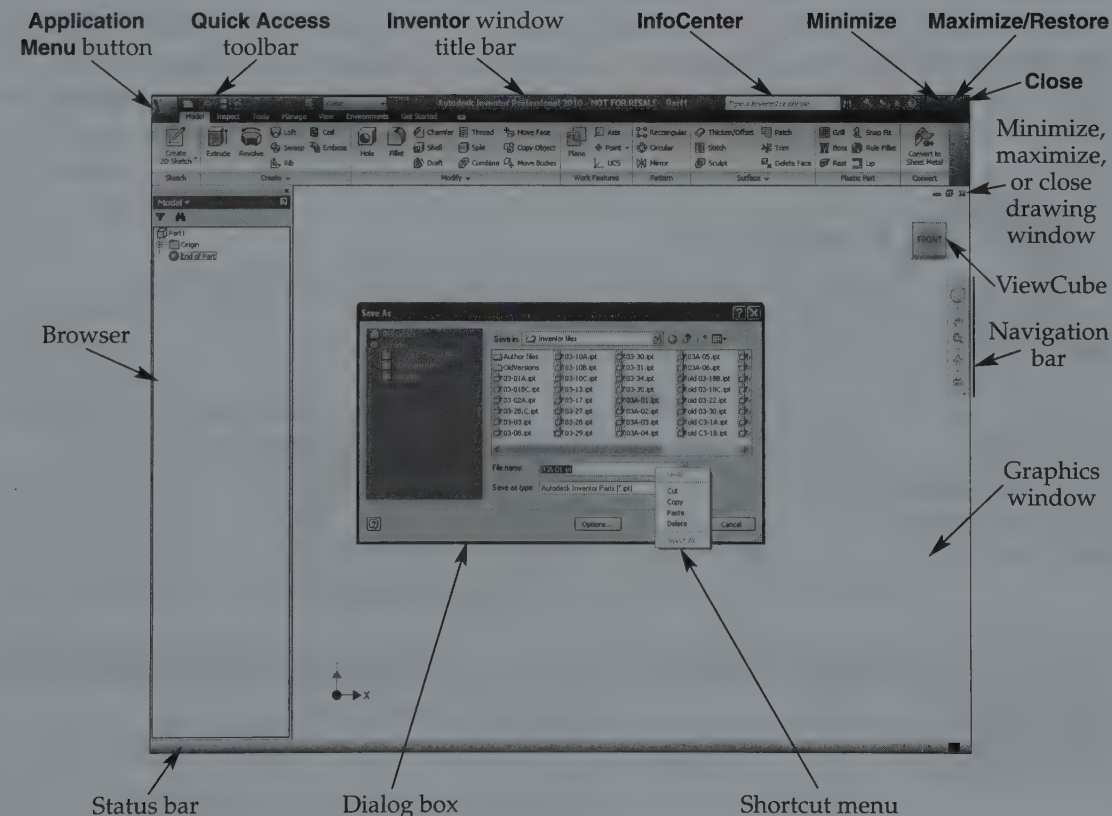
Application Menu button · Quick Access toolbar · *Inventor* window title bar · InfoCenter · Minimize · Maximize/Restore · Close · Minimize, maximize, or close drawing window · ViewCube · Navigation bar · Graphics window · Browser · Status bar · Dialog box · Shortcut menu

Cursor

The cursor is the primary means of pointing to locations within the Inventor window and making selections. The cursor changes to a crosshairs or other unique appearance when certain tools are active. Many visual cues, options, and tools are available from or associated with the cursor.

Tooltips

A *tooltip* displays when you hover over most tools or options. See **Figure 1-20**. Tooltip content varies depending on the item. The initial tooltip might only display the tool name, a *command alias* or *keyboard shortcut*, and a description. As you continue to hover, an explanation on how to use the tool and other information may appear.

Shortcut Menus

Inventor uses *shortcut menus* to simplify and accelerate tool and option access. If you right-click while a tool is active, the shortcut menu contains *context-sensitive menu options*. See **Figure 1-21**. Hover over a menu option that includes a small arrow to the right of the option name to display a *cascading menu*.

A tool or option accessible from a shortcut menu may appear as a graphic in the margin of this textbook, or directly within a sentence. The graphic represents the process of right-clicking, followed by selecting a menu or cascading menu option. You must right-click on an appropriate location within the correct work environment to access a specific option. The example shown in this margin illustrates accessing the sketch **Line** tool from the shortcut menu that appears when you right-click in the graphics window or browser in the sketch work environment, when no other tool is active.

tooltip: A pop-up that provides information about the item over which you are hovering.

command alias: Abbreviated tool name entered at the keyboard.

keyboard shortcut: Single key or key combination used to quickly issue a command or select an option.

shortcut menus (cursor menu, right-click menu, pop-up menu): Menus that allow access to tools and options by right-clicking anywhere in the graphics window or on a specific object or selection.

Shortcut

Create Line

context-sensitive menu options: Options specific to the tool currently in use.

cascading menu: A secondary menu that contains options related to the chosen menu item

Exercise 1-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-6.

Figure 1-20.
Place the cursor over a button to reveal the button name as a tooltip.
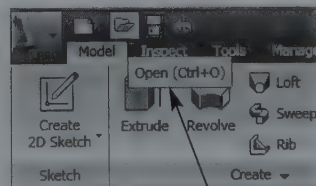
Tooltip for the **Open** button

Figure 1-21.
The shortcut menu that appears depends on the item under the cursor when you right-click.
Cascading menus are available from certain shortcut menus and flyouts.

Pick to repeat
the previous tool

Cascading
menu

Pick to access
help files

Right-clicked

**Right-Click in the
Graphics Window**

**Right-Click in the
Browser**

**Right-Click on a
Model Item**
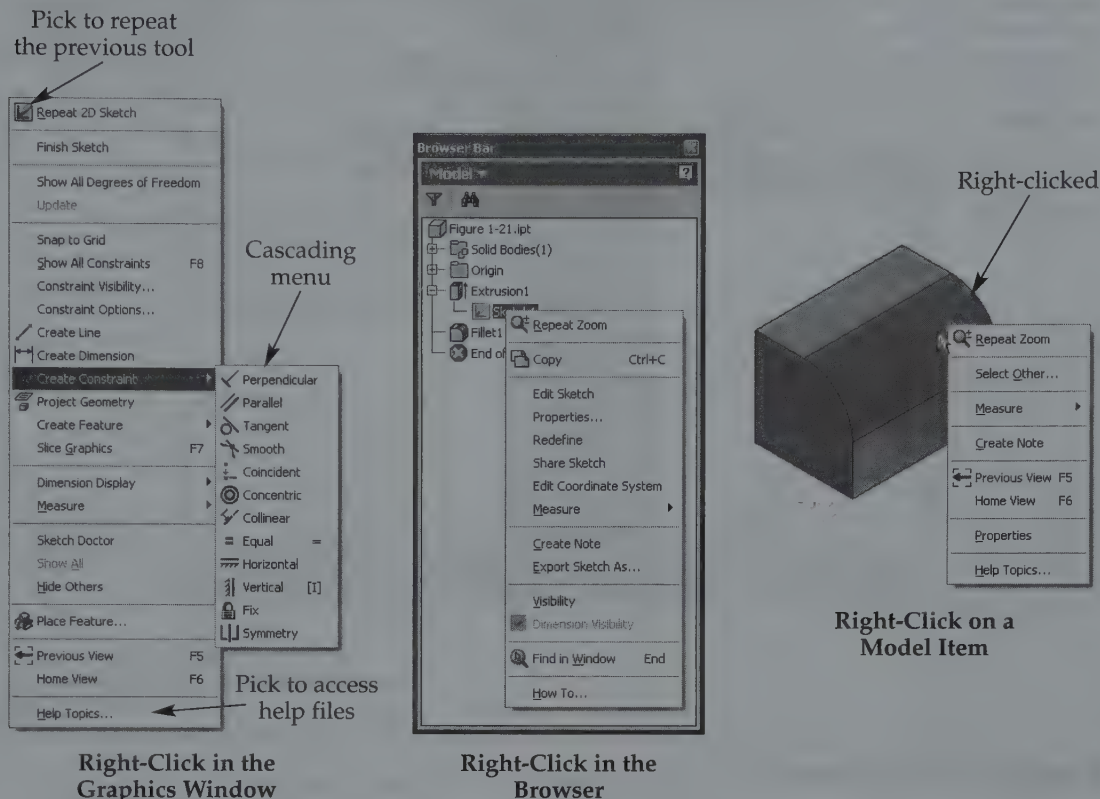
Application Menu

Pick the **Application Menu** button to display the **Application Menu**. See **Figure 1-22**.
The **Application Menu** provides access to program- and file-related tools and settings
through a menu system. Some items on the left side of the **Application Menu** function
as buttons to activate common application tools, and except for the **iProperties** button,
also display a menu. For example, press the **New** button to access the **New File** dialog
box.

To display a menu, hover the cursor over the menu name, pick the arrow on the
right side of the button, or select a menu button that does not activate a tool. Long
menus include small arrows at the top and bottom for scrolling through selections.
Pick an option to activate the tool. To exit the **Application Menu** without selecting an
option, pick outside of the **Application Menu** or press [Esc].

A tool or option accessible from the **Application Menu** appears as a graphic in the
margin of this textbook. The graphic represents the process of picking the **Application Menu**
button, followed by selecting a menu button, or hovering over a menu and picking a
menu option. The example shown in this margin illustrates accessing the **Save Copy
As Template** tool from the **Application Menu** as shown in **Figure 1-22**.

> **Application Menu**
> **Save As**
>
> **Save Copy As
> Template**

NOTE

Once a menu appears, you can use arrow keys to move to different
menu items and press [Enter] to select the highlighted item.

Figure 1-22.
Use the **Application Menu** to access common application and file management tools and settings, search for commands, and view open and recently used documents.

Application Menu button

Hover to display menu, or pick to activate tool

Search tool

Menu name

Pick to activate tool

Pick to exit inventor

Pick to display the **Application Options** dialog box
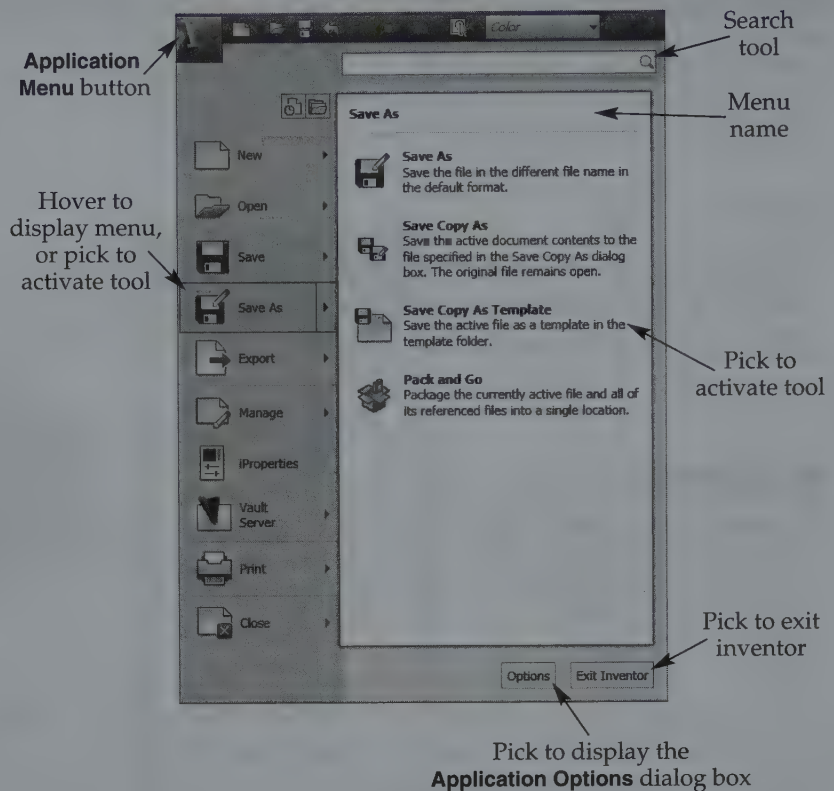
Searching for Commands

The **Application Menu** contains a search tool that allows you to locate and access a *command* available in the current file. For example, you must open a part model file to search for commands associated with part modeling. Type characters in the **Search** text box that might be included in the command name. Matches appear as you type. Typing additional letters narrows the search, with the best-matched command listed first. **Figure 1-23** shows using the **Search** text box to locate the **Save** tool for saving a file. Pick a command from the list to start the command.

Opening Recent Documents

Pick the **Recent Documents** button in the **Application Menu** to display a list of recently opened documents. See **Figure 1-24**. The **Recent Documents** function provides convenient access to files that are likely to be related to the current project. Select **By Ordered List**, **By Access Date**, **By Size**, or **By Type** from the **Ordered List** drop-down list to arrange recent files. Choose the appropriate option from the display options flyout to display files as icons, small images, medium images, or large images.

Hover over a file in the list of recent documents to display a tooltip with file information and a preview. Pick a file from the list to open. Pick the pushpin icon to the right of the file to keep the file on the recent documents list. Unpinned files are eventually removed from the recent documents list as you open other files.

Exercise 1-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-7.

Figure 1-23.
Use the **Application Menu** to search for a command. Pick the command from the list to activate it.

Pick to re-enter search characters

Type the command name

Pick to activate the tool

Matches for: "save"

Manage
Save — Manage > Styles and Standards > Save

AEC Exchange
Save As DWG Solids — AEC Exchange > Manage > Save As DWG Solids

Save
Save — Save > Save

Save All — Save > Save All

Save As
Save As — Save As > Save As

Save Copy As — Save As > Save Copy As

Save Copy As Template — Save As > Save Copy As Template

Options   Exit Inventor

Figure 1-24.
Use the **Application Menu** to browse through and select from recently opened files.

Pick to manage open documents

Select an option from the flyout to specify how to list recent files

Select an option from the flyout to specify how to display recent files

Pick to select recently opened documents

Recent Documents

By Ordered List

New
Open
Save
Save As
Export
Manage
iProperties
Vault Server
Print
Close

Chamfer   Thread   Move Face
Shell   Split   Copy Object
Draft   Combine   Move Bodies
Modify

Engine Piston.ipt
Clutch Gear 25t.ipt
Engine Case Rear.ipt
Coupling Zip-Tie.ipt
Engine Head.ipt
Flywheel Washer.ipt
Rear Exhaust - 2 Shoe

Pick to pin the file to the list

Pick to open

C:\Program Files\Autodesk\Inventor 2010\Samples\Models\Assemblies\Engine MKII\Components\Engine Head.ipt

Tooltip with preview image

Date modified: 2/12/2009 1:13:44 PM
Version info:
Last saved by:
Currently open by:

Quick Access Toolbar

The **Quick Access** toolbar, shown in **Figure 1-25**, is the only default toolbar. The **Quick Access** toolbar displays *tool buttons* that provide convenient access to common tools and options. As you move the cursor over a tool button, the button becomes highlighted and may display a border. Use the tooltip to become familiar with the tool icon, and then pick a tool button or select an option from a flyout to activate the corresponding tool.

When no file is open, the **Quick Access** toolbar offers enabled **New**, **Open**, and in some cases **Redo** tool buttons. Additional tool buttons appear when you open a file. There is some variation in available tools, or the top-level flyout option, depending on the active file type and current work environment.

A tool or option accessible from the **Quick Access** toolbar appears as a graphic in the margin of this textbook. The graphic represents the process of picking a **Quick Access** toolbar button, or an option from a flyout. The example shown in this margin illustrates accessing the **Redo** tool from the **Quick Access** toolbar.

> **NOTE**
>
> The **Quick Access** toolbar is fully customizable by adding, removing, and relocating tool buttons. Use options in the **Customize Quick Access Toolbar** flyout or right-click on a tool button to make basic adjustments.

Exercise 1-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-8.

Ribbon and Ribbon Panels

The ribbon, shown in **Figure 1-26**, is the primary means of accessing most design, drafting, and system tools and options. Use the *tabs* along the top of the ribbon to access related *ribbon panels*, or *panels*. For example, the **Model** ribbon tab in the part work environment includes panels with tools for creating and modifying part model sketches and features. Each panel houses groups of similar tools.

The ribbon adjusts to the current file type, work environment, and design or drafting task by providing context-sensitive tabs and panels. You can only use tools and options appropriate to the current conditions; all others are disabled. Highlighted tabs or panels include tools specific to the current work environment. For example, when you enter the sketch work environment to create a part model sketch, a highlighted **Sketch** tab appears and all tabs receive a highlighted **Exit** panel. Refer again to

Figure 1-25.
Use the **Quick Access** toolbar to access common tools. Pick a tool button to initiate the corresponding tool.

Flyout

No File Open

Pick to make basic toolbar adjustments
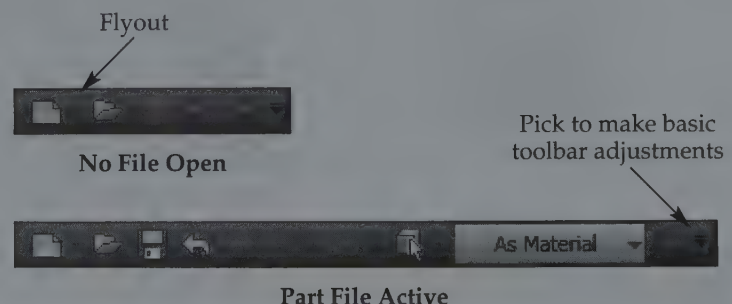
Part File Active

Figure 1-26.
The ribbon provides an effective, context-sensitive means of accessing most design, drafting, and system tools and options.
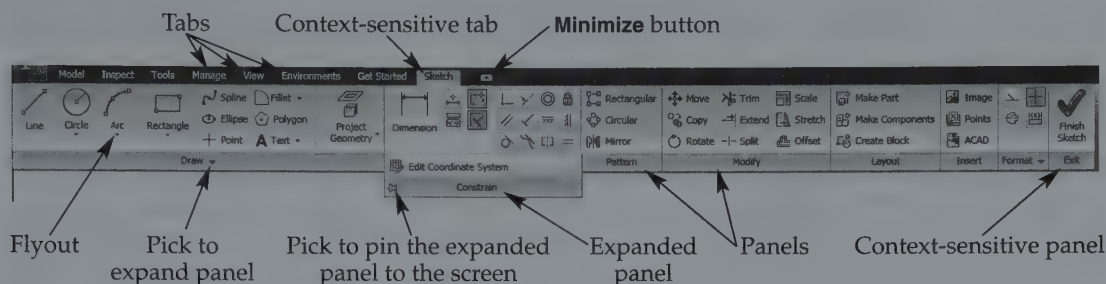

Figure 1-26. The **Sketch** tab includes tools and options for creating sketches, and the **Exit** panel includes a button for finishing and exiting the sketch. Tools that you cannot use in the sketch environment are disabled.

PROFESSIONAL TIP

Use the tabs displayed on the ribbon to help recognize the current work environment, and as a guide to proceed through the design and drafting stages.

A tool or option accessible from the ribbon appears as a graphic in the margin of this textbook. The graphic identifies the tab and panel where the tool is located. You may need to expand the panel or pick a flyout to find the tool. The example shown in this margin illustrates accessing the sketch **Line** tool from the **Draw** panel of the **Sketch** tab.

The large tool buttons in a panel signify the most often used panel tools. In addition to tool buttons, panels can contain flyouts and other items. Some panels have a triangle, or arrow, next to the panel name. If you see this arrow, pick the bottom, or title, of the panel to display a set of additional related tools and functions, as shown in **Figure 1-26**. To show the expanded list on-screen at all times, pick the pushpin button.
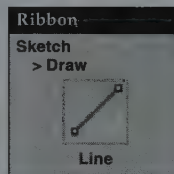
NOTE

When you pick an option from a ribbon flyout, the option becomes the new default and appears in the ribbon. This makes it easier to reselect the same tool.

Adjustments

Right-click on the ribbon, away from a panel, to access a shortcut menu with a variety of ribbon display options, as described in **Figure 1-27**. Most of the options are also available by right-clicking on a panel. By default, the ribbon *docks* horizontally below the Inventor window title bar. Use the **Undock** shortcut menu option to change the ribbon to a *floating* state, as shown in **Figure 1-28**. Resize the floating ribbon using the resizing arrows that appear when you move the cursor over the ribbon edge. Right-click as indicated in **Figure 1-28** to make additional floating ribbon adjustments.

To reposition a ribbon tab, hold down the left mouse button on a tab and drag the tab right or left, if the ribbon is in a horizontal orientation, or up or down, if the ribbon is in a vertical orientation. Release the button at the desired location. To reposition a panel within a tab, hold down the left mouse button on a panel title and drag the panel to the desired location.

dock: Set in position on an edge of the Inventor window (top, bottom, left, or right).

floating: Describes interface items displayed within a border that can be freely resized or moved.

Figure 1-27.
Right-click options for displaying and organizing ribbon elements.

Option	Description
Ribbon Appearance	Displays a submenu with options for turning off ribbon text and reducing ribbon size. Pick **Reset Ribbon** to remove custom ribbon settings.
Minimize	Provides a submenu with options for displaying the default full ribbon, tabs only, or panel titles only. Minimize options can also be activated by repeatedly pressing the **Minimize** button to the right of the ribbon tabs.
Panels	Displays a submenu of panels that you can add to or remove from the active ribbon tab. Checked panels appear in the tab.
Show Panel Titles	Check to display panel titles.
Customize User Commands...	Displays the **Customize User Commands** dialog box, which allows you to add commands to or remove commands from a custom **User Commands** panel in the active ribbon tab.
Undock Ribbon	Changes the ribbon to a floating state.
Docking Positions	Provides a submenu with options for docking the ribbon in the default top position or to the left or right of the main graphics area.

Figure 1-28.
Floating the ribbon is an example of basic interface customization.
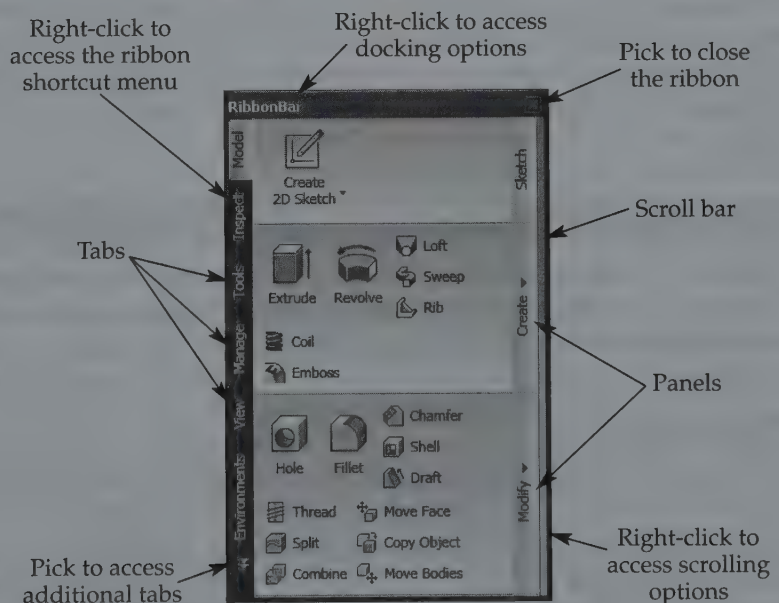
sticky panel: A ribbon panel moved out of a tab and made to float in the drawing window.

grab bars: Bars that appear at the edge(s) of a docked or floating item; used to move the item.

Drag a panel from a tab and drop it in the graphics window to create a *sticky panel*. See **Figure 1-29**. A sticky panel conveniently remains on-screen when you select a different ribbon tab. Hover over a sticky panel to reveal *grab bars* for moving the panel and buttons for returning the panel to its appropriate ribbon tab and adjusting the orientation of the title. You can also drag and drop a sticky panel back into a ribbon tab.

Figure 1-29.
A sticky panel created by dragging the **Create** panel from the **Model** tab and dropping it into the graphics window.

Pick to expand the panel

Pick to return the panel to its default location in the ribbon

Pick to change the location of the title and expansion button

Hold down mouse button and drag to move the panel to a new location

---
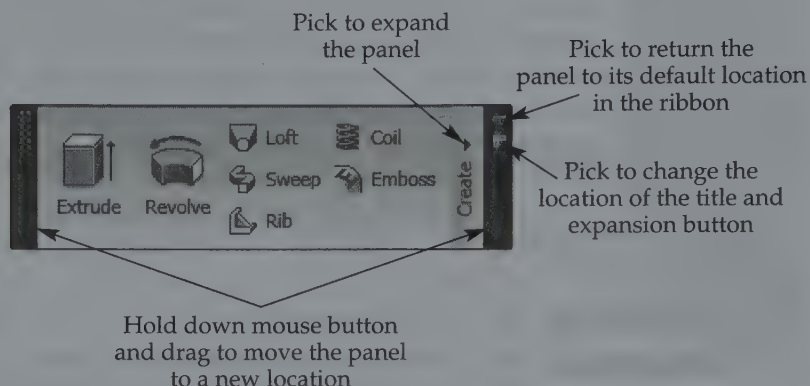
NOTE

The **Application Menu**, **Quick Access** toolbar, and ribbon replace traditional menu bars, toolbars, and the panel bar. You can customize the interface as desired, including displaying traditional interface items.

Exercise 1-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-9.

Keyboard Keys

An alternative method to access some Inventor tools and options is to press keys on the keyboard, known as *shortcut keys* or *keyboard shortcuts*. Some tools initiate when you to press a single key, while others require that you press a key combination. Keyboard keys provide quick access to certain tools, but they require you to memorize key or key combination functions.

Whenever it is necessary to cancel a tool or dialog box, press the *escape key* [Esc]. You may have to press [Esc] two or more times to cancel the operation. *Function keys* provide instant access to tools. Control and shift key combinations require that you press and hold the [Ctrl] or [Shift] key and then press a second character. You can initiate several tools using [Ctrl] combinations. A tooltip typically indicates if a key combination is available.

A tool or option accessible by typing appears as a graphic in the margin of this textbook, like the example shown in this margin. The graphic in the example shown in this margin represents the process of pressing the **E** key on the keyboard to access the **Extrude** tool.

shortcut key (keyboard shortcut): Single key or key combination used to issue a command or select an option.

escape key: Keyboard key used to cancel a tool or exit a dialog box.

function keys: The keys labeled [F1] through [F12] along the top of the keyboard.

Type ____
E

NOTE

Even though you will not be able to access some tools by typing keyboard shortcuts, you must still enter values at the keyboard. For example, you may have to enter the diameter of a circle or type a note using the keyboard.

Supplemental Material

Keyboard Shortcuts
For a list of default keyboard shortcuts, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Keyboard Shortcuts**.

Browser

browser (browser bar): A panel that displays all the items in the current model or drawing.

The *browser*, or *browser bar*, provides a historical reference of file content. For example, the part file browser displays all elements of the part model in the order in which you create the items. See **Figure 1-30**. The number and type of items available in the browser vary depending on the current file, work environment, and design stage. You can make changes to file content directly in the graphics window or from the browser.

parent node: An item in the tree structure, similar to a folder, that is associated with subordinate child nodes.

child node: Subordinate nodes that create, are associated with, or are consumed by a parent node item.

A tree structure, with *parent nodes* and *child nodes*, arranges many of the items in the browser. For example, a sketched feature is a parent node, and the sketch used to

Figure 1-30. The browser contains information regarding the content of a model or drawing. This browser is associated with a part file that contains two sketched features and a placed feature.

Solid bodies included in the model

The **Origin** folder contains planes, axes, and a center point to position and orient geometry

Pick to collapse the branch

Parent node

Child node

Pick to expand the branch

create the feature is a child node. The **Origin** folder is another example of a parent node. To display child nodes, pick the **Expand** button (+ symbol) to the left of the item name or right-click on the item name and select **Expand All Children**. To hide child nodes, pick the **Collapse** button (– symbol) to the left of the item name or right-click on the item name and select **Collapse All Children**. An element in the browser is *active* when all other items display a gray background.

> **NOTE**
>
> You can move, resize, float, and dock several Inventor interface items, including the Inventor and file windows and the browser. Different options and functions are available, depending on the particular interface item and floating or docked state.

Status Bar

The status bar appears along the bottom of the Inventor window. See **Figure 1-31**. The left side of the status bar provides information about a tool option by displaying a *help string* or prompt. For example, when you access the **Line** tool to sketch a line, the prompt states Select start of line, drag off endpoint for tangent arc.

help string: A short text description of what happens if you select a tool or option over which the cursor is hovering; or, if a tool is active, a prompt indicating the appropriate action.

Panes on the right side of the status bar indicate the number of occurrences in the document, number of open documents, and memory usage. A coordinate display field or the constraint information shown in **Figure 1-31** appears while you are sketching.

> **PROFESSIONAL TIP**
>
> Pay close attention to the prompts displayed at the status bar.

Exercise 1-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-10.

Dialog Boxes

Inventor uses *dialog boxes* throughout the design and drafting process. A dialog box appears whenever you pick a button displaying an ellipsis (…). Dialog boxes also commonly appear when you access a tool. Dialog boxes contain many of the same elements found in other interface items, including icons, text, buttons, and flyouts. **Figure 1-32** shows examples of two dialog boxes with several common dialog box elements.

dialog box: A window-like part of the user interface that contains various kinds of information and settings.

Figure 1-31.
The status bar displayed when a part file is open.

| Help string | Sketch resources | Number of occurrences | Number of open documents | Displays memory usage |

For Help, press F1

Fully Constrained   1   1
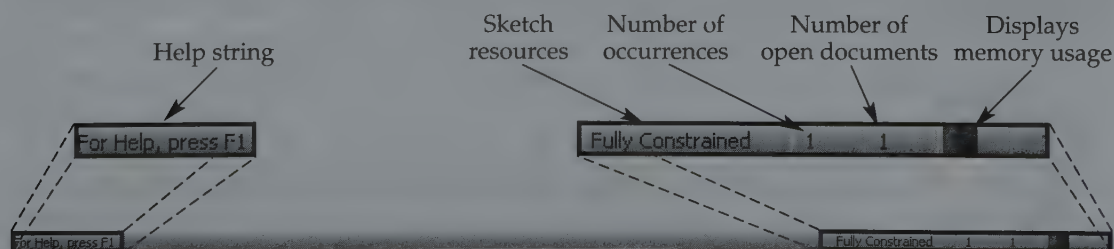
Figure 1-32. Pick a menu option or tool button that includes an ellipsis (…) to display a dialog box.

Drop-down list

Check box

Radio button

Text box

Command button

Tabs
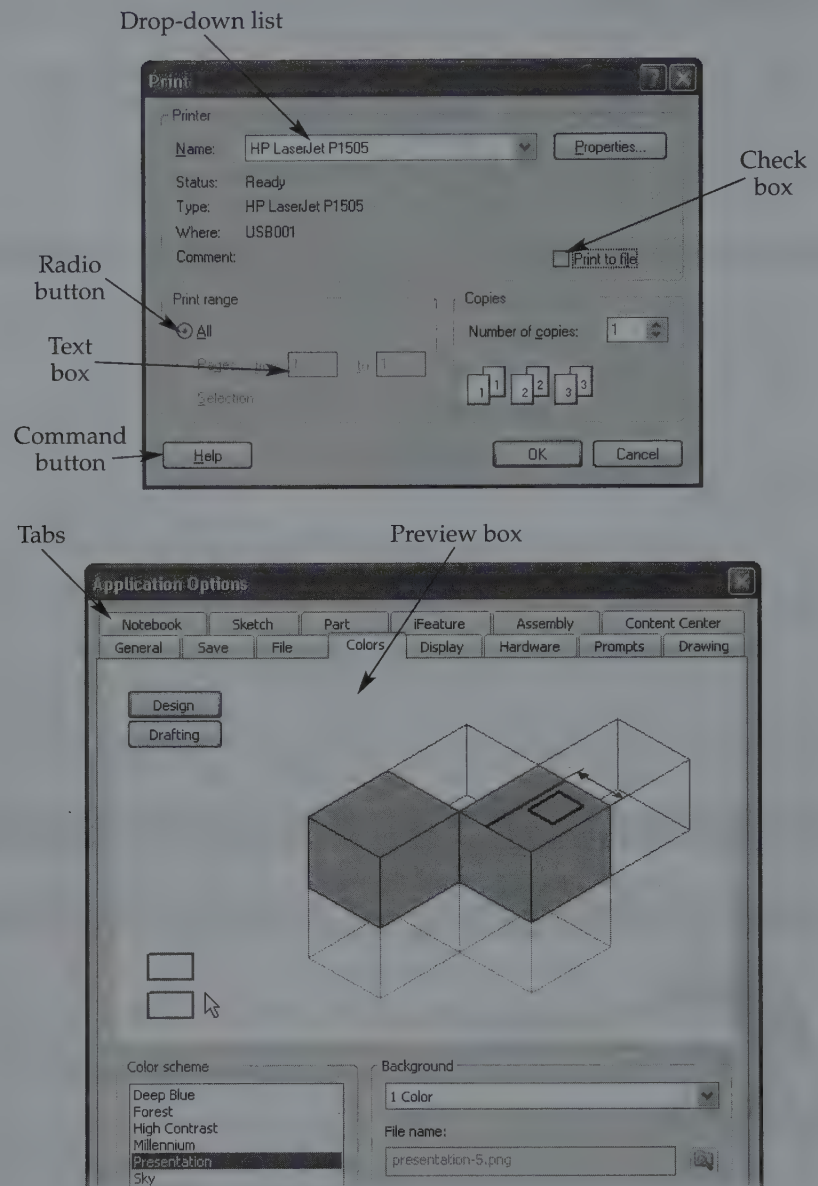
Preview box

preview box: An area in a dialog box that displays a preview of selected changes or options.

Use the cursor to set items in a dialog box. Many dialog boxes include images, *preview boxes,* or other information to help you to select appropriate options. Enter characters in text boxes to specify specific values. When you pick a button in a dialog box that includes an ellipsis (…), another dialog box appears. You must make a selection from the second dialog box before returning to the original dialog box. A button with an arrow icon requires you to select in the graphics window.

NOTE

You must complete an operation in a dialog box or cancel the dialog box to continue working.

Exercise 1-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-11.

Managing Multiple Documents

Most projects include a number of files and multiple file types. Each file presents or organizes a different aspect of the project. Documents associated with a project usually relate closely to each other. This is especially true when you are documenting a project with Inventor, because files reference other files. By opening multiple files at the same time, you can easily work between files and reference information contained in related documents.

Window Fundamentals

Each file you start or open in Inventor appears in its own window. The file name displays on the window title bar if the window is floating. The active file name appears on the Inventor window title bar. Standard Windows operating system window controls are available for adjusting the Inventor and file windows. To minimize, maximize, or close the Inventor window or file windows, pick the appropriate icon in the upper-right corner. You can also adjust the Inventor window by right-clicking on the title bar and choosing from the window control menu. Window sizing operations are also the same as those for other windows within the Windows operating system.

> **NOTE**
>
> Use the **Save All** tool available from the **Save** menu of the **Application Menu** to save changes to all open files. Use the **Close All** tool available from the **Close** menu of the **Application Menu** to close all open files.

Document Tabs

When you open two or more files, a document tab corresponding to each file appears above the status bar. See **Figure 1-33**. Document tabs are often the best way to manage and work between open files. Files are arranged in the order opened, with the file opened first on left side of the row. Drag a tab to the left or right of another tab to change the initial order. If you open so many files that tabs spread past the screen, use the **Scroll Right** and **Scroll Left** buttons that appear to the right of the tabs to access open files.

The highlighted tab identifies the active file. Hover over a tab to display an image of and path to the file. Pick a tab to make the file window active. You can also pick the **Open Documents** flyout, selected in **Figure 1-33**, to display, preview, and access an open file. Right-click on a tab to access options for closing, restoring, minimizing, maximizing, saving, and accessing iProperties of the associated file. This is a convenient way to mange individual files without first activating a file. Select the **Cascade** button to arrange open files in a cascading format. See **Figure 1-34A**. Pick the **Arrange** button to arrange open files in a tiled format. See **Figure 1-34B**.

Figure 1-33.
Use document tabs to manage multiple open files.

Select to arrange files in a cascading format

Select to arrange files in a tiled format

Pick to select a file from the flyout

Pick to close the active file

Pick to make the file active

Preview image

File path
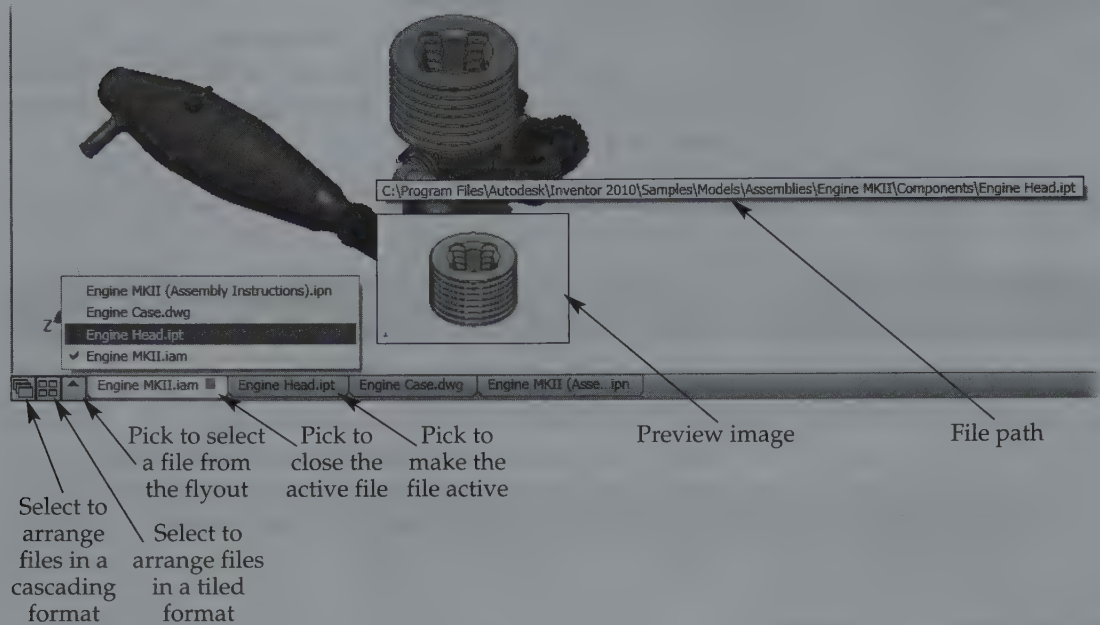
Additional Window Control Tools

The **Windows** panel of the **View** ribbon tab also provides options for viewing open files. Pick the **Switch Windows** flyout to display a list of all open files. Select a file from the flyout to make active. The **Tile** and **Cascade** buttons control the arrangement of open drawings, as shown in **Figure 1-34**.

Pick the **Open Documents** button in the **Application Menu** to display a list of open files in the **Application Menu**. Files are listed numerically in the order opened. You can display open files as icons, small images, medium images, or large images by picking the appropriate option from the display options flyout. To activate a different file window, pick the file from the list.

> **PROFESSIONAL TIP**
>
> Another effective technique for switching between open drawings is to press [Ctrl]+[F6].

check box: A selectable box that turns an item on (when checked) or off (when unchecked).

> **NOTE**
>
> Turn interface items on and off using the appropriate close function, or use the *check boxes* available from the **User Interface** flyout in the **Windows** panel of the **View** ribbon tab.

Exercise 1-12

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-12.

Figure 1-34.
A—Arranging open windows using the **Cascade** option. B—Arranging open windows using the **Tiled** option.

Active file   Pick to activate   Pick to   Pick to
                                                                   minimize   maximize
                                                                              Pick to
                                                                              close

A

Pick to   Active
activate   file
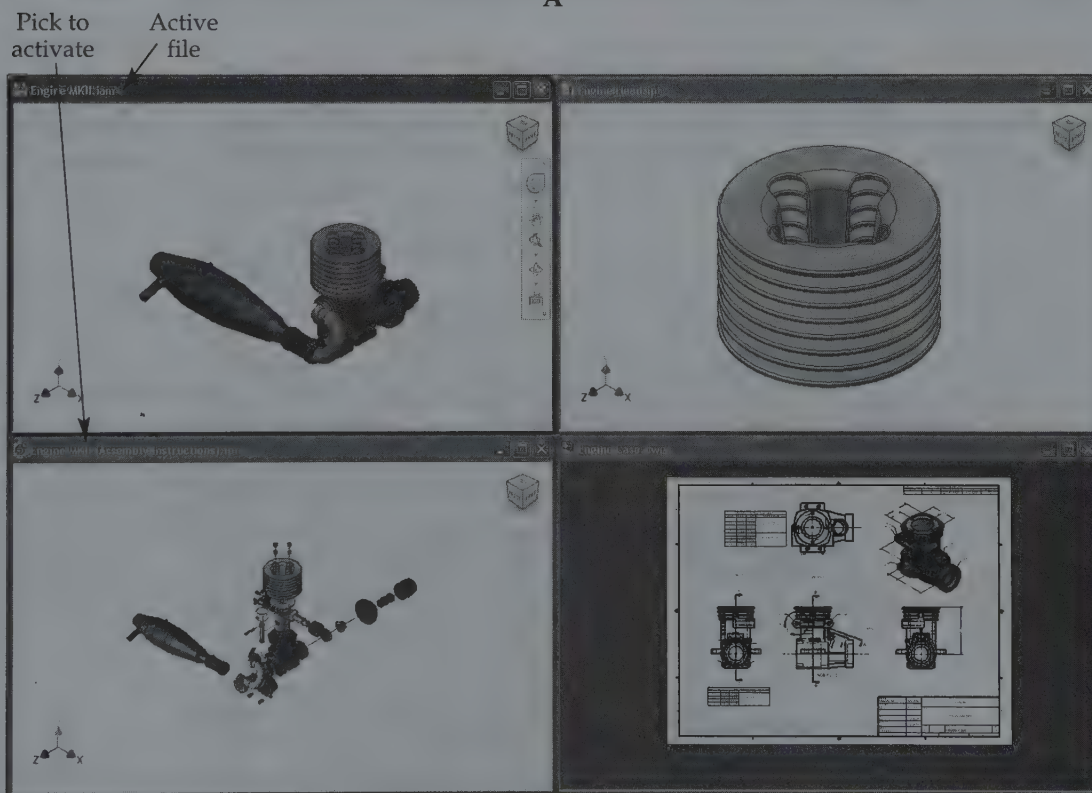
B

Application Options

Inventor system options are contained in the **Application Options** dialog box. See **Figure 1-35**. System options apply to the entire program and are not specific to a file. However, many application options help configure specific work environments and design and drafting tasks. For example, options in the **Sketch** tab control functions specific to preparing sketches.

Application options control an extensive number of Inventor functions. For example, redefine your username in **User name** text box of the **General** tab. Inventor references the username to identify who has worked on a file. Another example in the **General** tab is the **Start-up action** check box. When selected, this check box allows you to choose to display the **Open** dialog box, display the **New File** dialog box, or immediately begin a new file using a selected template when you launch Inventor.

As you learn Inventor, you will become familiar with many application options, and you may need to adjust the settings to fit your needs. You should explore the **Application Options** dialog box and learn to recognize the purpose and usefulness of each option. This textbook describes specific application options when applicable.

Figure 1-35.
The **General** tab of the **Application Options** dialog box. The **Application Options** dialog box controls many general, file-specific, and environment-specific program settings.

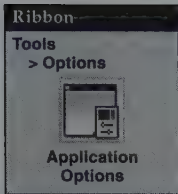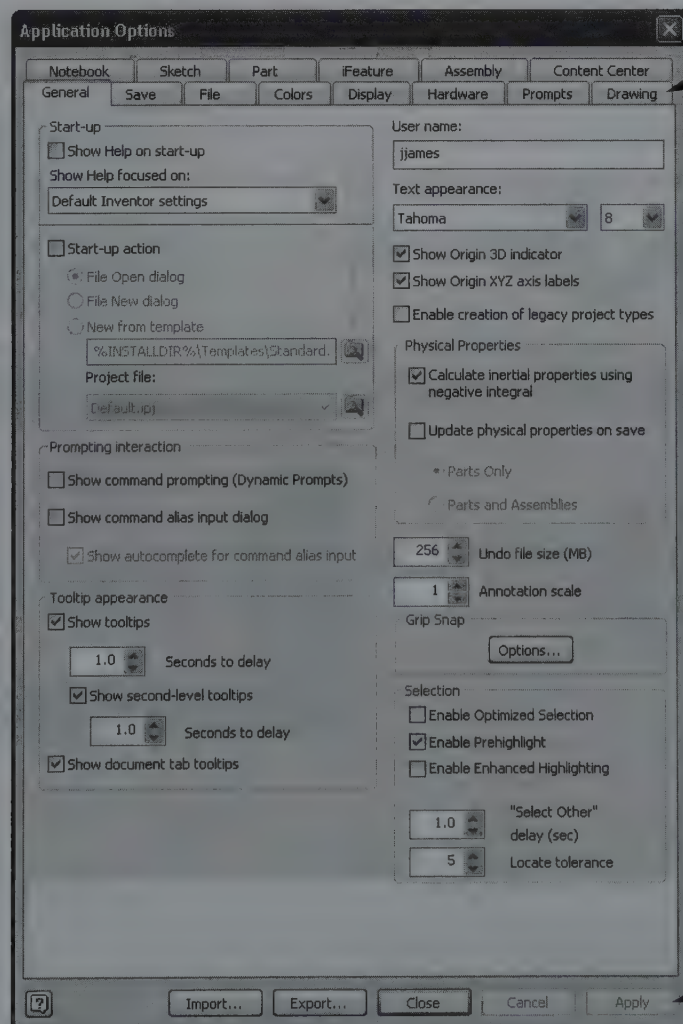

Pick a tab to access specific application options

Pick to apply options before closing

Exercise 1-13

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 1-13.

Getting Help

If you need help with a specific tool, option, or Inventor feature, use this textbook as a guide, or reference the help system contained in the **Autodesk Inventor Help** window. See **Figure 1-36**. In addition to the methods shown in the margin, you can access the **Autodesk Inventor Help** by pressing the [F1] key or picking the **Help** icon that is present in many dialog boxes. The help file system is similar to other Windows-based help files.

Shortcut
Help Topics...
InfoCenter

?
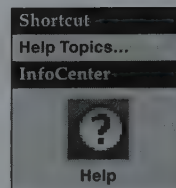
Help

PROFESSIONAL TIP

If you press the [F1] key while you are in the process of using a tool, help information associated with the active tool displays. This *context-oriented help* saves valuable time, since you do not need to scan through the help contents or perform searches to find information.

context-oriented help: Help information for the active tool.

InfoCenter

The **InfoCenter**, located on the right side of the application title bar and shown in **Figure 1-36**, allows you to search for help topics using the text box, without first displaying the **Autodesk Inventor Help** window. The **InfoCenter** also provides buttons for access to the **Subscription Center**, **Communication Center**, and **Favorites** list.

Figure 1-36.
Get help using the **Autodesk Inventor Help** window and **InfoCenter**.
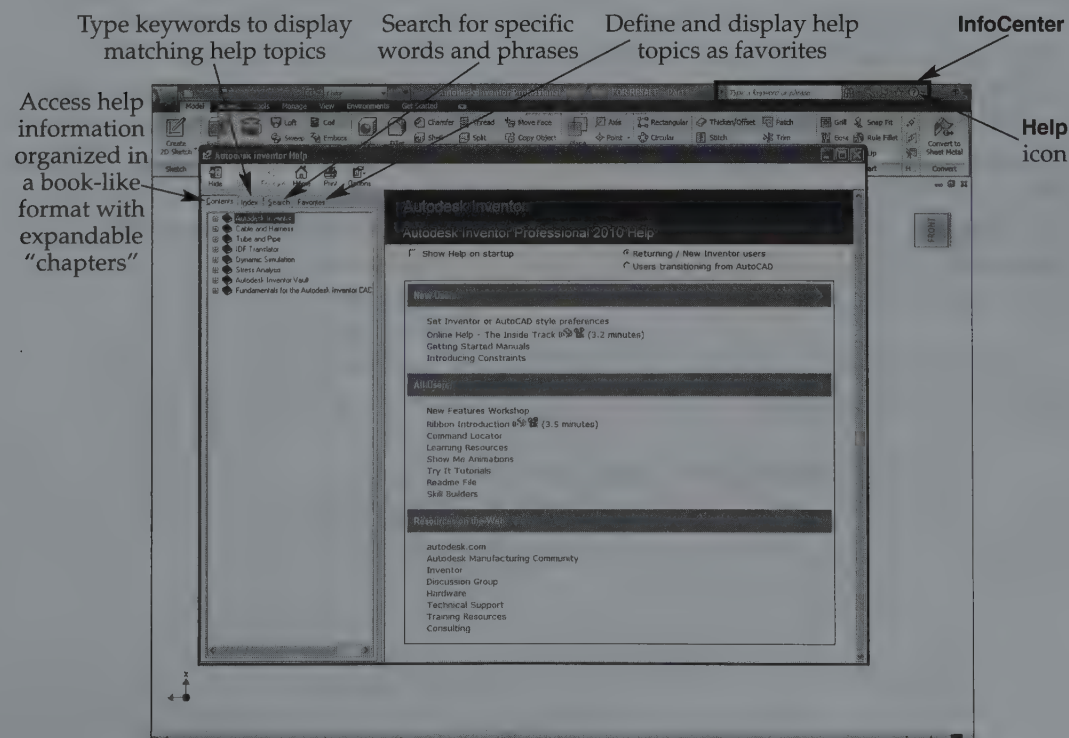
Type keywords to display matching help topics

Search for specific words and phrases

Define and display help topics as favorites

InfoCenter

Access help information organized in a book-like format with expandable "chapters"

Help icon

Chapter Test

1. What does the acronym CADD mean?
2. Define *solid model*.
3. How is a drawing different from a solid model?
4. What are drafting standards?
5. What is another name for tools?
6. Name the four basic file formats used in Inventor.
7. What are components?
8. Briefly describe parametric design and drafting.
9. What are geometric constraints?
10. How are dimensional constraints different from geometric constraints?
11. What does it mean to say a design is fully constrained?
12. What is the purpose of a sketch?
13. How do placed features relate to sketches?
14. What are work features?
15. What does it mean to hover?
16. Define *default*.
17. What term identifies a set of related buttons that appears when you pick the arrow next to certain tool buttons?
18. Briefly describe how to open and save a new file.
19. When and why does an alert display?
20. Briefly describe the Inventor user interface.
21. Describe the graphics window.
22. What is a command alias?
23. What name is given to small stubs at the top or side of a page, window, dialog box, or palette, allowing access to other portions of the item?
24. What is the ribbon?
25. What term describes interface items that are locked into position on an edge of the window (top, bottom, left, or right)?
26. What is the basic function of the [Esc] key?
27. What term describes the keys labeled [F1] through [F12] along the top of the keyboard?
28. What two window control tools allow you to organize multiple files so they can all be seen and accessed in the graphics window?
29. Briefly explain how to change Inventor system options that apply to the entire program.
30. How can you open the **Autodesk Inventor Help** window?

Problems

Launch Inventor and complete the activities described in each problem.

1. Perform the following tasks:
   A. Begin a new Inventor assembly file using a template of your choice.
   B. Explore the **Application Menu** by picking any of the menus and moving the cursor down and up through the menu options.
   C. Pick the **Application Menu** and highlight the **New** menu.
   D. Pick the **Open** button in the **Application Menu** to display the **Open** dialog box.
   E. Press the [Esc] key to close the **Open** dialog box.
   F. Pick the **Options** button to access the **Application Options** dialog box.
   G. Pick the **Close** button to exit the **Application Options** dialog box.
   H. Close the assembly file without saving.
2. Perform the following tasks:
   A. Begin a new Inventor part file using a template of your choice. Explore the default part interface.
   B. Close the part file without saving.
   C. Begin a new Inventor assembly file using a template of your choice. Explore the default assembly interface.
   D. Close the assembly file without saving.
   E. Begin a new Inventor drawing file using a template of your choice. Explore the default drawing interface.
   F. Close the drawing file without saving.
   G. Begin a new Inventor presentation file using a template of your choice. Explore the default presentation interface.
   H. Close the presentation file without saving.
3. Perform the following tasks:
   A. Begin a new Inventor part file using a template of your choice.
   B. Locate and explore the **Quick Access** toolbar.
   C. Place the cursor over several toolbar buttons and observe the tooltip and help strings provided.
   D. Pick one of the flyouts located in the **Quick Access** toolbar and select one of the flyout options to initiate a tool.
4. Perform the following tasks:
   A. Open blade_main.ipt from the following folder:
      Autodesk/Inventor 2010/Samples/Models/Assemblies/Scissors/Components
   B. Right-click on the ribbon and pick **Small** from the **Ribbon Appearance** cascading menu. Right-click on the ribbon and pick **Compact** from the **Ribbon Appearance** cascading menu. Right-click on the ribbon and pick **Text Off** from the **Ribbon Appearance** cascading menu. Right-click on the ribbon and pick **Normal** from the **Ribbon Appearance** cascading menu.
   C. Locate and explore the default browser.
   D. Drag the **End of Part** indicator up and above Sketch4 and notice the changes.
   E. Pick the **Expand [+]** button to the left of the **Origin** folder to expand all children.
   F. Right-click on the **Origin** folder and select **Collapse All Children**.
   G. Close blade_main.ipt without saving.

5. Write a brief description of the Inventor interface. Include answers to the following questions: What is an interface? What are the primary Inventor interface items, and how do they function? Reflect on the concept of a computer software interface. Are some of the Inventor interface items similar to those in other programs you have used? Submit a hard copy of your description to your drafting instructor or supervisor.

6. Write a short description of each of the four Inventor file types. Include information about file extensions and describe when you should use each file type. Reflect on the capabilities of Inventor. How is Inventor similar and different from other CADD programs you may have used? Submit a hard copy of your description to your drafting instructor or supervisor.

7. Write a brief report on each part model element and the importance of constraints and parametric associations in parametric design and drafting. Reflect on your understanding of parametric modeling, and describe your experience with working in parametric situations. Submit a hard copy of your report to your drafting instructor or supervisor.

8. Create a new part file and draw a freehand sketch of the standard Inventor screen display. Label each of the screen areas and the interface items described in this chapter.

Type = Single User

Location = C:\Documents and Settings\jjames\[

Included file =

Use Style Library = Read Only

Workspace

Workgroup Search Path

Libraries

  Samples - C:\Program Files\Autodesk\Inventor 2010\Samples

Frequently Used Subfolders

CHAPTER

Design Preparation

2

Learning Objectives

After completing this chapter, you will be able to do the following:
- ✓ Create and edit design project files.
- ✓ Access and create template files.
- ✓ Control document settings.
- ✓ Specify iProperties.
- ✓ Explain the purpose of styles and standards.

Before you begin creating models and drawings, you should recognize the tools and options that assist in organizing and setting up a *design session*. Proper planning and use of specific file preparation and management tools and techniques will increase your success and productivity. This chapter explains setup functions that are important for planning and completing a design.

design session:
Time spent working on a project, including analyzing design parameters and using Inventor.

Projects

Inventor requires that design stages and documents within a design associate with each other. **Figure 2-1** shows the file structure for a small design project and illustrates file referencing. The exploded assembly drawing file, for example, references a presentation file, which references an assembly file, which references three part files. Inventor must maintain links between files to enable proper file referencing and preserve parametric relationships. An assembly file, for example, only references component files; the assembly file does not house the components. If you move or delete the component files, the assembly is unable to reference the needed files.

Figure 2-1.
An example of the file structure and references that occur between files for a design project.
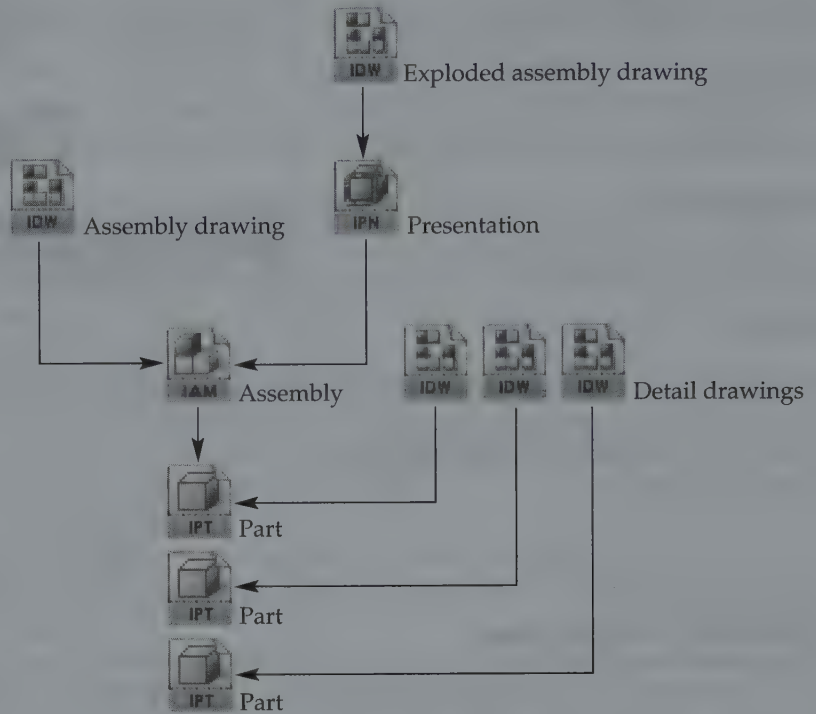
Exploded assembly drawing

Assembly drawing

Presentation

Assembly

Detail drawings

Part

Part

Part

Projects help organize connections between files and aid in the design process. Projects are separate files that carry the .ipj extension. Create and use as many projects as necessary to manage your work in an efficient manner. Typically, you should develop a project for each unique job or design project. A project is a virtual or electronic form of a real project, such as the guitar shown in **Figure 2-2**. The guitar design consists of parts, subassemblies, an assembly, technical illustrations, and a complete set of working drawings. The project created specifically for the guitar design contains and manages each part, assembly, presentation, and drawing file associated with the guitar.

Figure 2-2.
Using a project to organize all folders, files, and relationships between files necessary to build a product, such as this guitar, will help you become an efficient and successful Inventor user. (Model courtesy of Ethan Collins)

Using the Projects Dialog Box

Use the **Projects** dialog box to create and adjust projects. See **Figure 2-3**. The **New File** and **Open** dialog boxes include a **Projects...** button for convenient access to the **Projects** dialog box. The upper pane displays loaded projects. The **Project name** column lists the name of each project, and the **Project location** column identifies the path to the corresponding project file. Click once on a project name to view and manage associated project files and settings in the lower pane. To activate the project, double-click on the project in the upper pane or select the project and pick the **Apply** button. A check mark to the left of the project name identifies the current project.

Ribbon
Get Started
> Launch

Projects

NOTE

The current project remains active until you activate a different project. An alert message appears if you open or save a file in or from a location not specified in the current project. You must close all Inventor files in order to activate a different project.

PROFESSIONAL TIP

Activate a project while using the **Open** or **New File** dialog box by selecting a project from the **Project file** drop-down list.
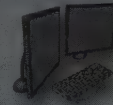
Exercise 2-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 2-1.

Figure 2-3.
This **Projects** dialog box allows you to create and manage Inventor projects.

Loaded projects

Active project

Move certain category items up

Move certain category items down

Add new path

Project settings

Open the **Choose project file** dialog box

Display all the files that have the same names in the project

Add, remove, import, or modify content center libraries
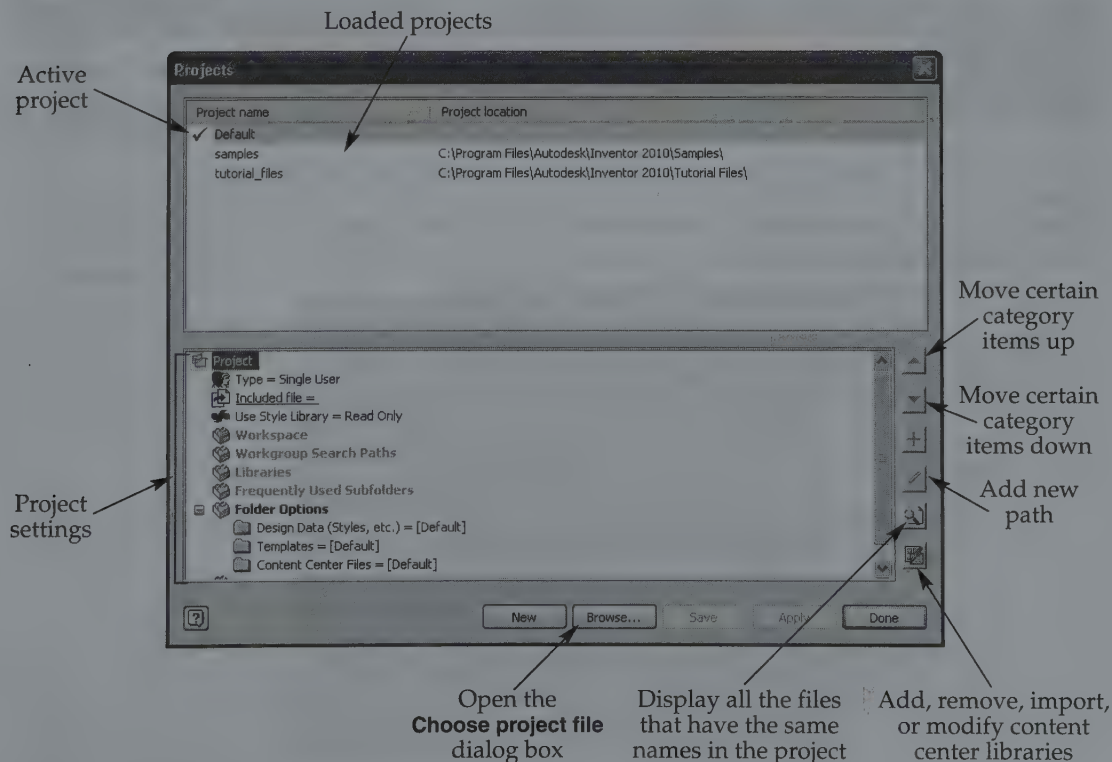
Creating a Project

To create a project, pick the **New** button or right-click on a project in the upper pane of the **Project** dialog box and select **New...** to display the **Inventor project wizard.** By default, and if the Autodesk Vault software is installed, the first page of the **Inventor project wizard** contains the options shown in **Figure 2-4**. Autodesk encourages the use of Vault to manage design team projects. This textbook focuses on the process of creating and using single-user projects without Vault. Pick the **New Single User Project** *radio button* to create a new single user project. Then pick the **Next** button to advance to the next page.

NOTE

Pick the **Cancel** button at any time to cancel the **Inventor project wizard**, or pick the **Back** button to return to the previous page.

The next page provides a **Project File** area to define the name and location of the project file. See **Figure 2-5**. Consider a project to be a resource that manages folders and files associated with a design project. For example, the designer of the guitar described previously may store all folders and files related to the guitar in a folder titled Guitar, which is stored in a folder such as My Documents on the C: drive. Inside the Guitar folder might be subfolders, such as Parts, Subassemblies, Presentations, and Drawings. The guitar project file coordinates the location of each folder in the Guitar folder, along with any other folders outside of the Guitar folder that are associated with the guitar design.

Enter a descriptive name for the project file in the **Name** text box. Then pick the ellipses (**...**) button to access the **Browse For Folder** dialog box. Use the dialog box to pick a folder, or *workspace*, to store project subfolders and files. To create a folder if a folder does not exist, pick the **Make New Folder** button. The project name is often the same as the name of the main folder in which you store subfolders and files, as specified in the **Project (Workspace) Folder** text box. For example, store the exercise and problem files you create in this textbook in an Exercises and Problems folder, which you manage using a project named Exercises and Problems.

NOTE

Projects can reference many folders and, as a result, do not need to share the name of a specific folder.

Figure 2-4.
Use the first page of the **Inventor project wizard** to specify the project type. This textbook focuses on the creation of single-user projects.
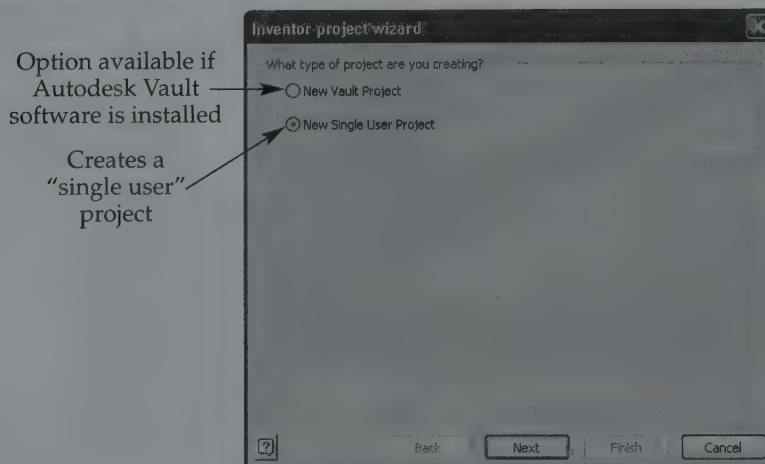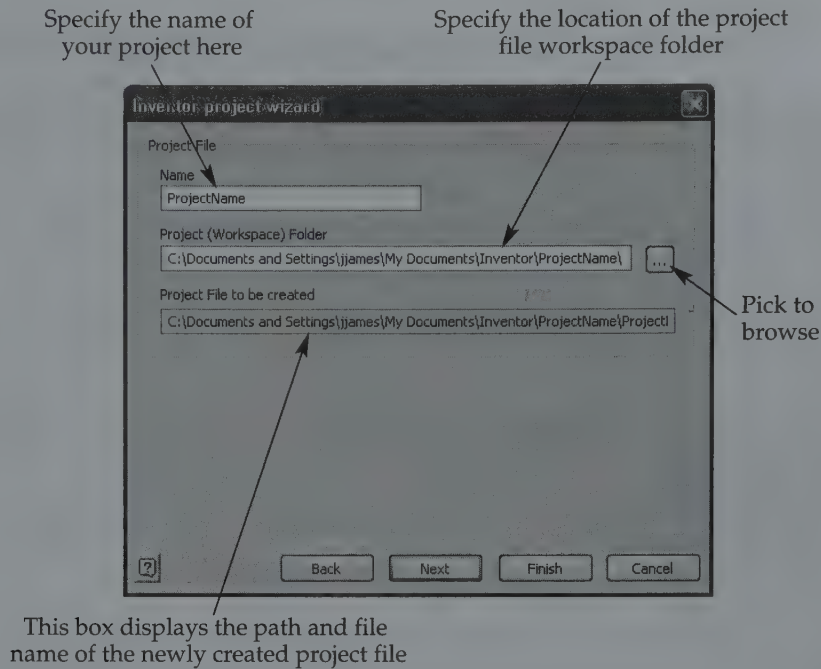
Option available if Autodesk Vault software is installed

Creates a "single user" project

Figure 2-5.
Use the second
page of the **Inventor
project wizard** to
assign a name to the
project and specify
its location.

Specify the name of
your project here

Specify the location of the project
file workspace folder

Pick to
browse

This box displays the path and file
name of the newly created project file

Once you specify the project name and location of project subfolders and files, pick the **Next** button to display the last page. See **Figure 2-6**. The last page allows you to assign *library search paths* to existing *libraries* for the project. Only libraries assigned to the current project are available. To add a library search path, select the libraries to add from the **All Projects:** list box and pick the **Add selected libraries** button. Pick the **Delete selected libraries** button to remove a library search path. After you complete each page of the wizard, pick the **Finish** button to create the new project file. The project appears in the upper pane of the **Projects** dialog box and is initially not active.

library search paths: Locations in which Inventor looks for library files on your computer or the network.

library: A folder that contains files used in a project or several different projects.
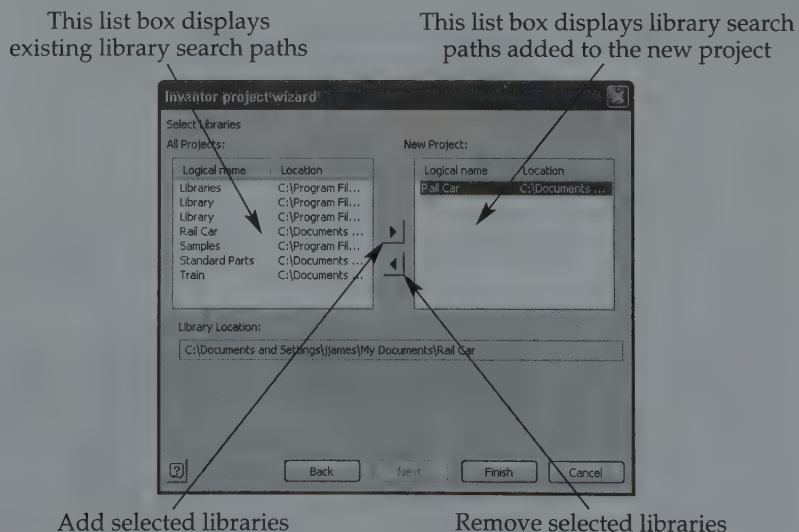
Exercise 2-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 2-2.

Figure 2-6.
The last page of
the **Inventor project
wizard** allows you
to add libraries
assigned to the
active project to the
new project.

This list box displays
existing library search paths

This list box displays library search
paths added to the new project

Add selected libraries

Remove selected libraries

Managing Projects

Right-click on a project in the upper pane of the **Projects** dialog box to access options for creating a new project and for renaming, locating, or deleting an existing project. The **Rename** and **Delete** options become enabled only when you right-click on an inactive project. Pick the **Rename** option to type a new name for the project. Select the **Delete** option to remove the project from the list without eliminating the project file.

Select the **Browse...** option to display an existing project that is not currently shown in the **Projects** dialog box. The **Choose project file** dialog box appears, allowing you to locate an existing project file. Select the **Open as read-only** check box to open the project file as *read-only*. Retrieving an existing project is common if you reinstall Inventor, delete the project from the list, or move to a different workstation.

> **PROFESSIONAL TIP**
>
> Even if you delete a project from the **Projects** dialog box, the project file remains until you delete the file from your computer. Use the **Browse...** option to retrieve such a project, instead of recreating the project.

The lower pane of the **Projects** dialog box displays a Project folder with folders or categories that specify how a project searches for files and operates. Use as few or as many project search paths and controls as needed, based on the location of required folders and files. Remember that the purpose of a project is to ensure that Inventor can efficiently find and reference files and design content and to help you work productively. **Figure 2-7** shows an example of a project search path structure. Refer to this figure as you learn the items often included in a project.

Expand a folder or category within the Project folder to view the specified folders or options. To make changes to a folder or category, or to the content of a folder or category, use the buttons available to the right of the lower pane, identified in **Figure 2-3**, or right-click on the item and select an option. **Figure 2-8** describes options you may find in a shortcut menu in addition to standard clipboard functions. Selecting a menu option may display text boxes for typing a value or path and a **Browse** button for locating a folder or file.

Figure 2-7.
Projects, project search paths, and project options can be confusing. Use this figure to help recognize project categories and understand single user project search paths.
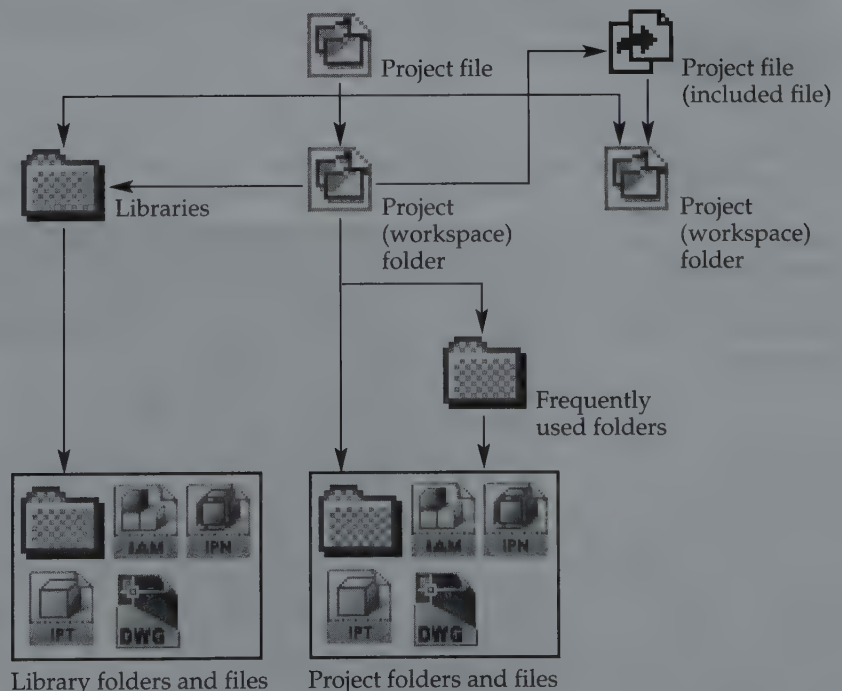

Library folders and files   Project folders and files

Figure 2-8.
Options you may find when you right-click on a project category, folder, or option. Some of the same functions are also available from the buttons to the right of the lower pane.

Option	Function
Edit	Adds a new search path or option or edits an existing one.
Open	Opens a project file for editing in the **Projects** dialog box.
Add Path	Adds a search path to a folder.
Add Paths from Directory	Provides the **Browse For Folder** dialog box to locate a search path.
Add Paths From File	Adds all search paths assigned to an existing project file to the selected project.
Add Proxy Path	Provides the **Browse For Folder** dialog box to locate a search path.
Paste Path	Pastes a copied or cut search path.
Move Up	Moves a folder up in the list so it will be searched sooner.
Move Down	Moves a folder down in the list so it will be searched later.
Delete	Removes the search path or option from the project.
Delete Selection Paths	Removes all libraries in a single operation.
Use Default Folder	Returns a folder to the default folder.

Included File

Add an *included file* to have access to search paths and information in another project file. You can add one included file to the project in order to *combine* the project file with the content of the included file.

included file: A separate project file linked to the current project.

Workspace

The **Workspace** folder is the location in which you keep most folders and files associated with a specific project. The workspace for the guitar project example is the Guitar folder previously described. A project does not require a workspace, but removing a workspace often defeats the purpose of using a project. The Default project is available without a workspace for applications in which a project is not necessary. When you use a project without a workspace, Inventor looks everywhere on your computer or the network to locate files. A project can include a single workspace.

Workgroup Search Paths

The **Workgroup Search Paths** category can include one or more workgroup folders. Workgroup search paths must be separate from the workspace. You typically assign workgroup folders to projects for locating files on a server or other computers on a network, primarily in a design team setting.

Libraries

The **Libraries** category can include one or more library folders. Add a library to a project to provide a link to common or shared files that are often used for multiple projects. An example of a library is a Screws folder that contains machine screw part files. Extensive library systems are common in mechanical design and drafting. For example, a folder named Fasteners might include separate folders named Screws, Washers, and Nuts, each containing many part files. Adding libraries to a project increases productivity

by making library components easily accessible and maintaining file references. Use a descriptive name for a library, usually the same name as the selected library folder.
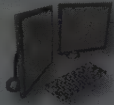
Saving standard components inside a workspace makes it difficult to locate and reference library files for use in other projects. Store standard parts and subassemblies in appropriate library folders, separate from the project, and then add the library search path to projects as needed.

Frequently Used Subfolders

frequently used subfolder: A virtual folder within a project that stores the paths to folders and files you use frequently.

The **Frequently Used Subfolders** category can include one or more *frequently used subfolders*. Use frequently used subfolders in the same way you use subfolders for folder and file organization. You must store frequently used subfolders in the workspace folder. For example, a project might include a frequently used subfolder named Parts, stored in the project folder, that contains all part files specific to the project.

Exercise 2-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 2-3.

Folder Options

The **Folder Options** category contains Design Data (Styles, etc.), Templates, and Content Center Files folders. These folders allow you to customize access to design data, including styles, templates, and content center files. Create your own folders and files within the folders, in a location of your choice, and then link those folders to a specific project. This allows default design data, templates, and content center files to remain unchanged, while providing you easy access to specific folders related to the project.

Supplemental Material

Project Options

For information about the **Options** category in the lower pane of the **Projects** dialog box, including options for saving old versions of the project and importing non-Inventor files, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Project Options**.

Manage projects outside of Inventor using the **Inventor Project Editor 2010** available by selecting the Windows Start button > Programs > Autodesk > Autodesk Inventor 2010 > Tools > Project Editor or by right-clicking on a project file and selecting **Edit**. The **Inventor Project Editor 2010** functions the same as the **Projects** dialog box, but it also contains an **Inventor** button to launch Inventor. Double-click on a project file in Windows Explorer to launch Inventor with the selected project active.

Opening and Saving Files

When you activate a project other than the Default project, the **Open**, **Save As**, **Save Copy As**, and all similar dialog boxes display the folder assigned to the current project by default, allowing you to locate folders and files associated with the active project. **Figure 2-9** illustrates the link between the current project and the **Open** dialog box. The link provides an effective means of working with project files without spending significant time navigating to file locations. The pane in the upper-left corner of the **Open** or **Save** dialog box provides convenient access to project categories, including libraries and frequently used subfolders. Pick the **Workspace** icon to navigate back

Figure 2-9.
The benefit of creating and using projects is apparent when you open or save files in the active project.
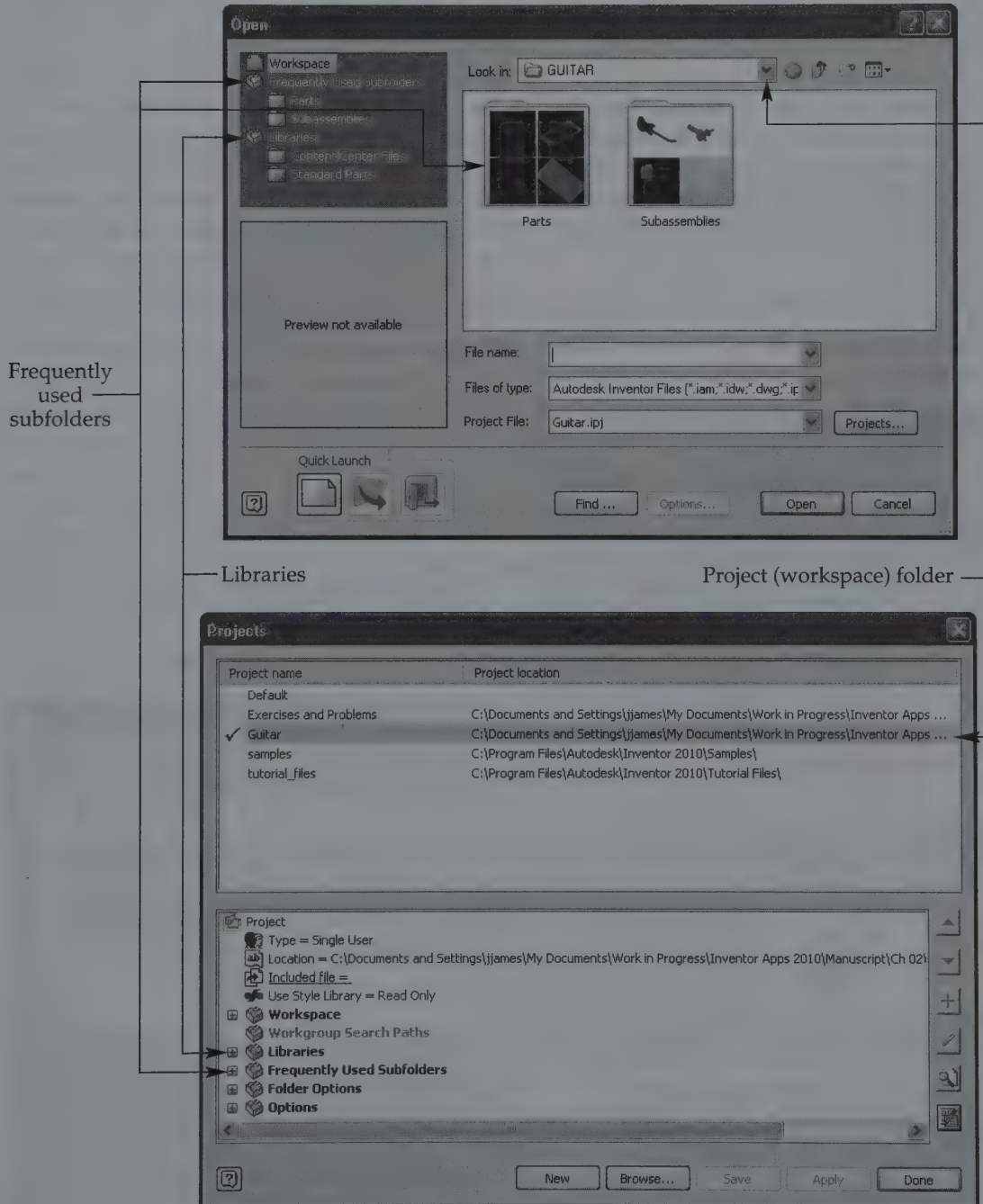
to the project (workspace) folder. Although the contents of the active project appear initially, you can still locate and open any file on your computer or the network as needed.
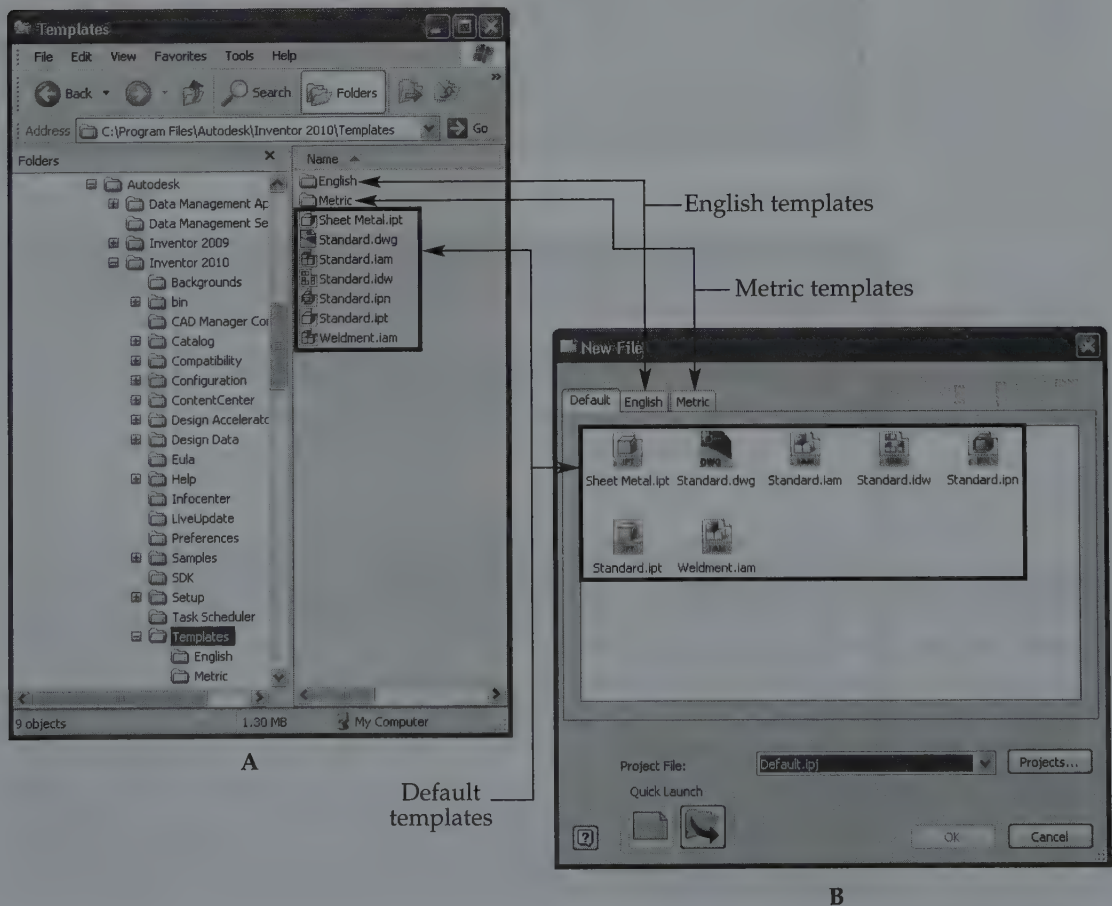
Templates

Template options and specifications vary depending on the file type, project, and design and drafting standards. For example, you might use a template for designing steel parts and another for designing aluminum parts, or a template for metric drawings and another for U.S. Customary (inch) drawings. As shown in **Figure 2-10**, the Autodesk/Inventor 2010/Templates path stores the default tabs and templates found in the **New File** dialog box. The Inventor-provided templates are adequate for many applications. However, to increase efficiency and maintain uniformity, you should create templates based on specific industry, school, and company standards.

A custom template can save time and effort because file settings you establish are preset each time you use the template to begin a new file. For example, part file templates include unit settings, sketch environment grid spacing, and color, material, and lighting styles. Drawing file templates include unit settings, sheet content such as a title block and border, and drawing standards. You can develop templates for each file type and prepare specific templates for sheet metal part and weldment modeling.

Figure 2-10.
A—The default location of Inventor templates. B—The corresponding display in the **New File** dialog box.

Assigning Templates to Projects

One of the best ways to organize and access templates is to assign a template folder to a project using the **Projects** dialog box or **Projects Editor**. This allows default templates and folders to remain unchanged, while providing access to custom or project specific templates. To assign a template folder to a project, access the **Projects** dialog box or **Projects Editor** and select the project to which you want to assign the template folder. Next, right-click on the Templates folder in the **Folder Options** category and pick **Edit**. Pick the **Browse** button to access the **Browse For Folder** dialog box and locate an existing template folder, or if you have not already created a folder, pick the **Make New Folder** button.

Give the templates folder a descriptive name, such as Templates, and save the folder in an appropriate location on your computer or the network. You typically should not save a templates folder in the project (workspace) folder, because many projects may need to reference the templates. Add folders to the templates folder as needed. For example, you might create an Inch folder for storing inch unit templates and a Metric folder for storing metric unit templates. **Figure 2-11** shows an example of a templates folder with two subfolders assigned to a project.
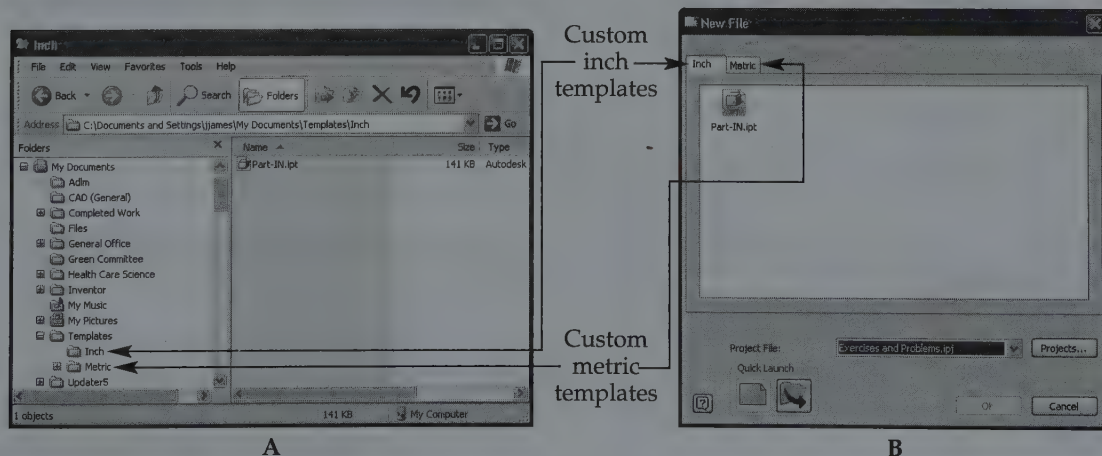
Creating a Template

You can use different methods to develop templates and create **New File** dialog box tabs. One of the most effective techniques for creating a new template is to begin a new file as a base for the template, followed by making changes to the file, and then saving the file using an appropriate template name and path. You can create a folder to display as a tab in **New File** dialog box while saving the template.

To apply this method, if you assigned a different template folder to the active project, make the Default project current so that you can access existing templates. Now, display the **New File** dialog box and start a new file using an available template file. For example, use Standard.ipt to create a part file template. Make changes to the file as needed to create custom template settings and then use the **Save As** dialog box to save the file. Give the template a descriptive name, such as Part-IN.ipt, and place the file in the templates folder assigned to the appropriate project. If necessary, you can create a new folder or subfolder by selecting the **Create New Folder** button. Finally, activate the project assigned to the template folder.

Now when you open the **New File** dialog box, the template tab and files you created are visible, as shown in **Figure 2-11**. A template appears in the **New File** dialog

Figure 2-11.
A—An example of a custom template folder with subfolders and a custom template file. B—The corresponding display in the **New File** dialog box.
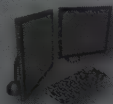
box only if the file exists in the associated template folder. A tab displays only if the corresponding template folder contains one or more files.

Exercise 2-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 2-4.

Document Settings

Ribbon

Tools
> Options

Document Settings

Document settings determine several general work environment characteristics associated with the current file. The **Document Settings** dialog box, shown in **Figure 2-12**, allows you to view and modify document settings. As you learn to use Inventor, you will discover how specific document settings apply to design and documentation. Once you understand the purpose of document settings, make changes to document settings in your template files as needed to preset the values each time you start a new file. This textbook describes specific document settings when applicable.

Units

document units: The units used to define linear, angular, time, and mass measurements and precision in models and drawings.

The **Units** tab allows you to set the file units of measurement, or *document units*. Document units are basic file settings that you should consider as an element of design preparation. **Figure 2-13** shows the most common units for inch and metric designs. Select the appropriate units of measurement for the file using the drop-down lists in the **Units** area. Choose suitable linear and angular precision values using the drop-down lists in the **Modeling Dimension Display** area. Specify how to display dimensional constraints by selecting the appropriate radio button. Chapter 4 describes dimensional constraints.

Exercise 2-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 2-5.

Figure 2-12.
The **Document Settings** dialog box contains options, often preset in template files, which control certain file or work environment characteristics.
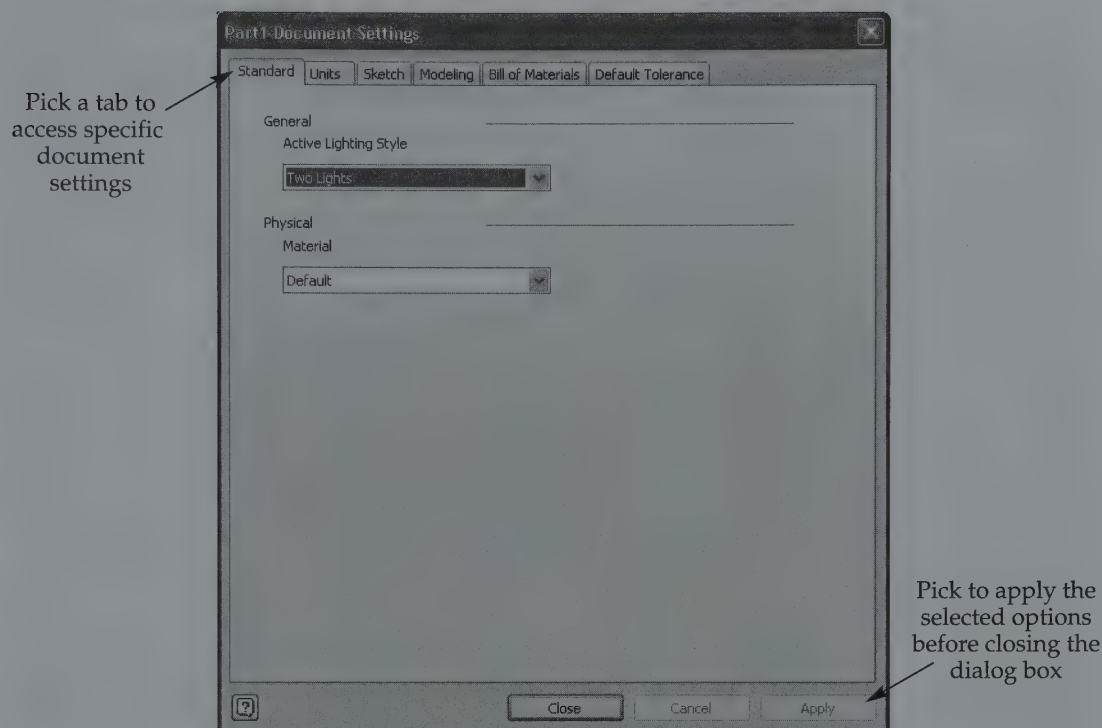

Pick a tab to access specific document settings

Pick to apply the selected options before closing the dialog box

Figure 2-13.
The most common units and precisions for inch and metric designs.

	Length	Angle	Time	Mass	Linear Precision	Angular Precision
Inch	Inch	Degree	Second	Lbmass	3.123	2.12
Metric	Millimeter	Degree	Second	Kilogram	1.1	2.12

iProperties

iProperties assign many important file specifications. iProperties range from the file type and name to the physical properties of the material used to manufacture the product. The **iProperties** dialog box allows you to view and assign iProperties to each file. See **Figure 2-14**. iProperties are critical throughout the design process to organize, manage, and document specific design and project data. A variety of applications, such as title blocks, parts lists, reports, and bills of materials collect and reference the information you specify in the **iProperties** dialog box.

iProperties:
Inventor file properties used to define a variety of file and design characteristics.

Application Menu

iProperties

NOTE

Open the **iProperties** dialog box in Windows Explorer or Inventor Design Assistant by right-clicking on an Inventor file and picking **iProperties**. Within Inventor, an alternate method to open the **iProperties** dialog box is to right-click on the name of the part in the browser and pick **iProperties**.

Figure 2-14.
iProperties and the content added to the **iProperties dialog box** are critical in certain parametric relationships. You will use this dialog box throughout the design process. The **General** tab of the **iProperties dialog box** lists general file information that you cannot modify directly in dialog box.

Pick a tab to adjust specific file properties

Pick to apply changes before closing the dialog box
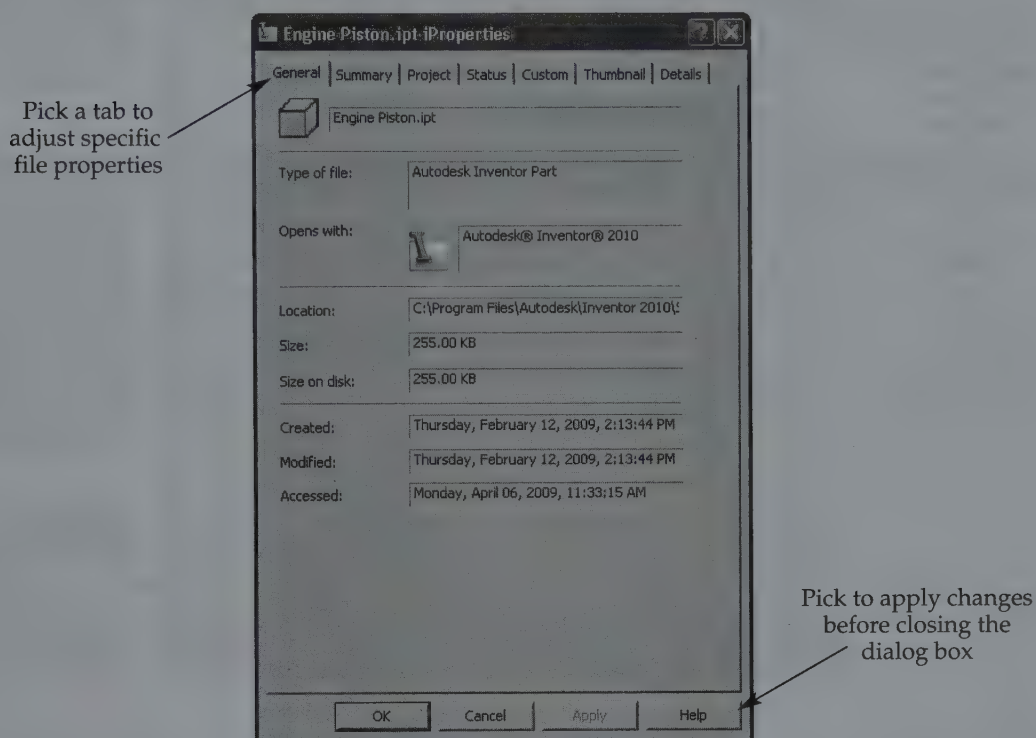
The **iProperties** dialog box contains tabs, each with specific file properties. The tabs include display, list, text, and check boxes; drop-down lists, radio buttons, and buttons for specifying properties. **Figure 2-15** provides a description of the contents of each tab. Systematically go through the **iProperties** dialog box and add the properties appropriate to each file. This textbook describes specific iProperties when applicable.

NOTE

The **Author** and **Designer** properties use your computer user name each time you begin a new file, regardless of the values you enter in a template. To use a different value for these properties, such as your initials, access the **General** tab of the **Application Options** dialog box and change the value in the **User name** text box.

PROFESSIONAL TIP

You can specify properties at any time during the design process. However, it is most effective to define properties when you begin to develop a design. Additionally, you can specify many common properties used throughout a project in your template files so that every time you create a new file, the properties that remain unchanged are already defined.

Figure 2-15.
Descriptions of the content in each tab of the **iProperties dialog box**. If you are working at your computer, access the **iProperties dialog box** and display each tab as you read the description.

Tab	Content
General	General file properties: name, type, location, size, and dates. You cannot use the **Properties** dialog box to modify general properties.
Summary	Properties that describe the file and product: title, subject, author and manager names or initials, company name, category, keyword, and comments.
Project	Properties associated with the file and project: part and stock numbers, description, revision number, project name or code, designer, engineer and authority names or initials, cost, creation date, product vendor, and Web address.
Status	Properties that organize and help control information related to file or product design status: design state, checks, approvals, and file status.
Custom	User-defined properties required to further organize and help control the file. Custom properties also allow you to add information that is not available from the default properties to title blocks, parts lists, reports, and bills of materials.
Save	Settings related to the file image displayed in the preview area of the **Open** dialog box. The **File Save Options** dialog box provides the same options.
Physical	Properties that specify product material and physical characteristics. Inventor analyzes the physical properties of a part or assembly file and shows how different materials and dimensions influence the physical and inertial properties of the model.
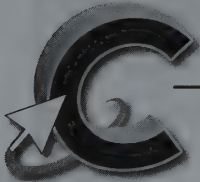
Exercise 2-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 2-6.

Styles and Standards

Models use styles to assign specific color, lighting, material, and model-specific characteristics. For example, if you build a model of a stainless steel bolt, you can use a stainless steel color, a specific lighting configuration, and a stainless steel material. Drawings use styles to control drawing display characteristics, including dimension and annotation styles and multiple other drawing elements.

Style Management

Inventor includes numerous default styles. If an available style is appropriate for your application, activate the style during the design or drafting process to assign the style to the file. If none of the default styles work for your application, create a copy of an existing style that is similar to your requirements and edit the new style as needed.
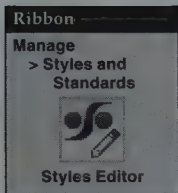
Save the new style in a template file to make the style available whenever you use the template to begin a file, or save the style to the style library to use the style in any file.

The Style and Standard Editor

Ribbon

Manage
> Styles and
**Standards**

Styles Editor

The **Style and Standard Editor** is the primary resource for observing and modifying the characteristics of existing styles and for creating and importing new styles. See **Figure 2-16**. The **Style and Standard Editor** contains a browser, a style area with multiple options, a filter drop-down list, and a number of buttons. Inside the browser are parent nodes associated with different styles.

You can use the **Style and Standard Editor** to specify a vast number of styles and settings. As you learn Inventor, you will become familiar with many styles applied to models and drawings. You may need to adjust the settings to fit design or project requirements. Explore the **Style and Standard Editor** and learn to recognize the purpose and usefulness of each style. This textbook describes specific **Style and Standard Editor** options when applicable.

Figure 2-16.
The **Style and Standard Editor** for a part or assembly controls color, lighting, and material styles. Notice the large number of color styles listed in the **Color Style** area. Sheet metal parts include styles specific to sheet metal, presentations offer color and lighting styles, and drawings include many drawing and annotation styles
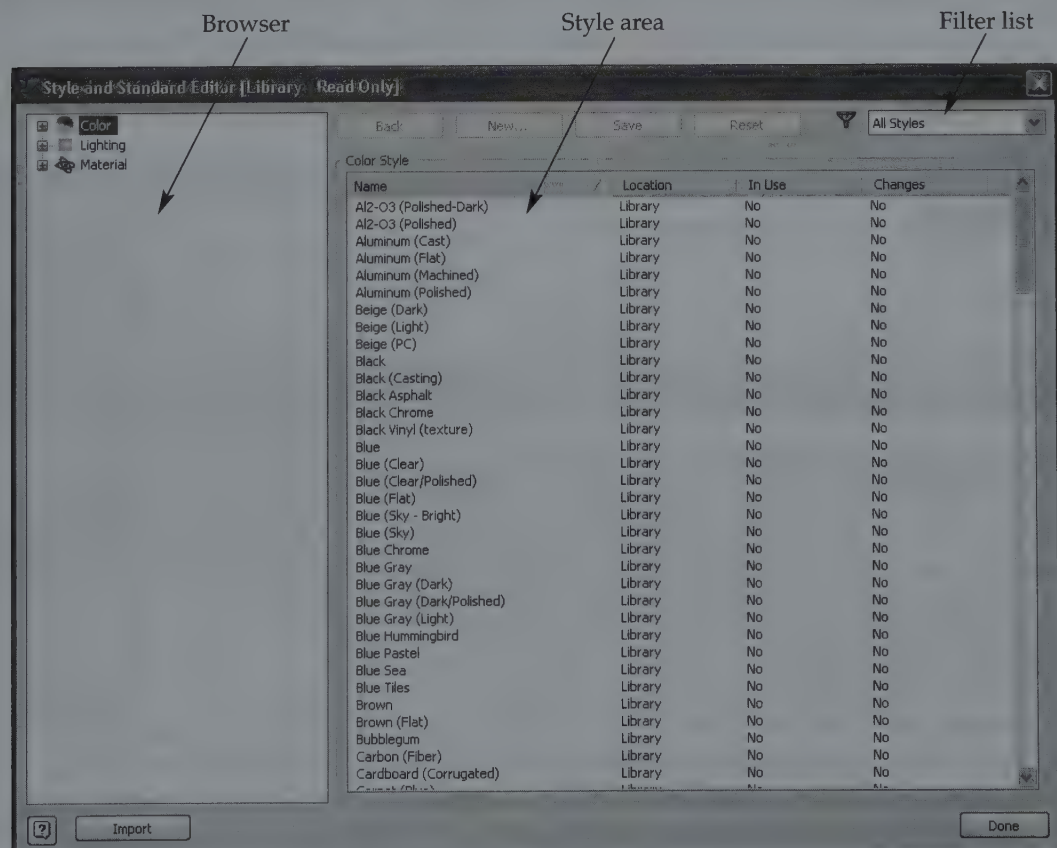
Browser   Style area   Filter list

Chapter Test

1. What are projects?
2. Explain the purpose of using projects in Inventor.
3. Define *workspace*.
4. What is a library?
5. Which libraries are available for a specific project?
6. Explain how to specify a library search path for a project.
7. Briefly describe how to create a new project.
8. Describe how to change the name of an existing project from within the **Projects** dialog box.
9. What is an included file?
10. Give an example of a library.
11. Explain how to ensure quick and easy access to frequently used folders within a project.
12. Explain the benefit of creating your own custom templates and assigning them to your projects using the **Projects** dialog box.
13. Why should you select a location other than your project folder to store your template files?
14. Define *document units*.
15. What unit of linear measurement is common in metric design files?
16. Briefly describe iProperties and explain what they represent.
17. Describe a process that would allow a company's engineering and manufacturing divisions to digitally "sign off" on a product design.
18. Explain how and where styles are used and give an example of when you might use a style to enhance a project.
19. What can you do if none of the styles provided with Inventor fit your needs for a particular project?
20. What is the primary resource for modifying existing styles and creating new styles?

Problems

Launch Inventor and complete the activities described in each problem. When you are finished, continue working with Inventor or exit if necessary.

1. Create a new single-user project named Problem 2-1. Place the project in a new folder named Problem 2-1, in a location somewhere on your hard drive or the network. Do not add an included file, libraries, or frequently used subfolders. Leave all options set as default.

2. Begin a new Inventor part file. Access and close the **Projects** dialog box, **Document Settings** dialog box, **iProperties dialog box**, and **Style and Standard Editor**.

3. Create a new single-user project named Problem 2-3. Place the project in a new folder named Problem 2-3, in a location somewhere on your hard drive or the network. Then perform the following tasks.
   A. Include your Exercises and Problems project file.
   B. Add a frequently used subfolder named Subfolder.
   C. Delete the project you just created from the list in the upper pane of the **Projects** dialog box.
   D. Use the **Browse** function to locate the deleted project and return it to the **Projects** dialog box list.

4. Write a brief description, in your own words, of the properties found in each of the **iProperties** dialog box tabs. Submit a hard copy of your description to your drafting instructor or supervisor.

5. **(REQUIRED)** Create a metric part file template. You will use the template throughout this textbook to create metric unit part models. The following steps guide you through the process of creating the template.
   A. Access the **Open** dialog box and select Default.ipj from the **Project file** drop-down list to activate the Default project. Pick the **Start a New File** button to load the Default project and display the default template tabs and templates.
   B. Start a new part file using the Standard (mm).ipt template on the **Metric** tab.
   C. Save the file as Part-mm.ipt in your Metric folder.
   D. Access the **Document Settings** dialog box and change the linear precision to 1.1. Confirm that linear units are set to millimeter, angular units are set to degree, time is set to second, mass is set to kilogram, and angular precision is set to 2.12, and that the **Display tolerance** radio button is selected.
   E. Pick **Apply** and close the **Document Settings** dialog box.
   F. Access the **iProperties** dialog box and type the appropriate initials, or your initials, in uppercase letters in the **Author**, **Manager**, **Designer**, **Engineer**, and **Authority** text boxes. Type the appropriate initials, or the initials of your instructor or supervisor, in uppercase letters in the **Checked By**, **Eng. Approved By**, and **Mfg. Approved By** text boxes. Type the name of your school or company in the **Company** text box. Type 0 in the **Revision number** text box. Type the Web address of your school or company in the **WEB link** text box. Leave everything else blank or set as default.
   G. Pick **Apply** and close the **iProperties** dialog box.
   H. Save and close the template file.
   I. Reactivate the Exercises and Problems project.

6. Write a brief report on the purpose and importance of Inventor projects. Identify a real-world design project and describe how you would use an Inventor project to organize and manage the files. Submit a hard copy of your report to your instructor or supervisor.

7. With the Exercises and Problems project active, access the **Open** dialog box and pick the **Samples** library to navigate to the **Samples** folder. Open the Suspension.iam file from the following folder: Samples/Models/Assemblies/Suspension. Draw a freehand sketch of what you see on-screen, not including the Inventor interface items.
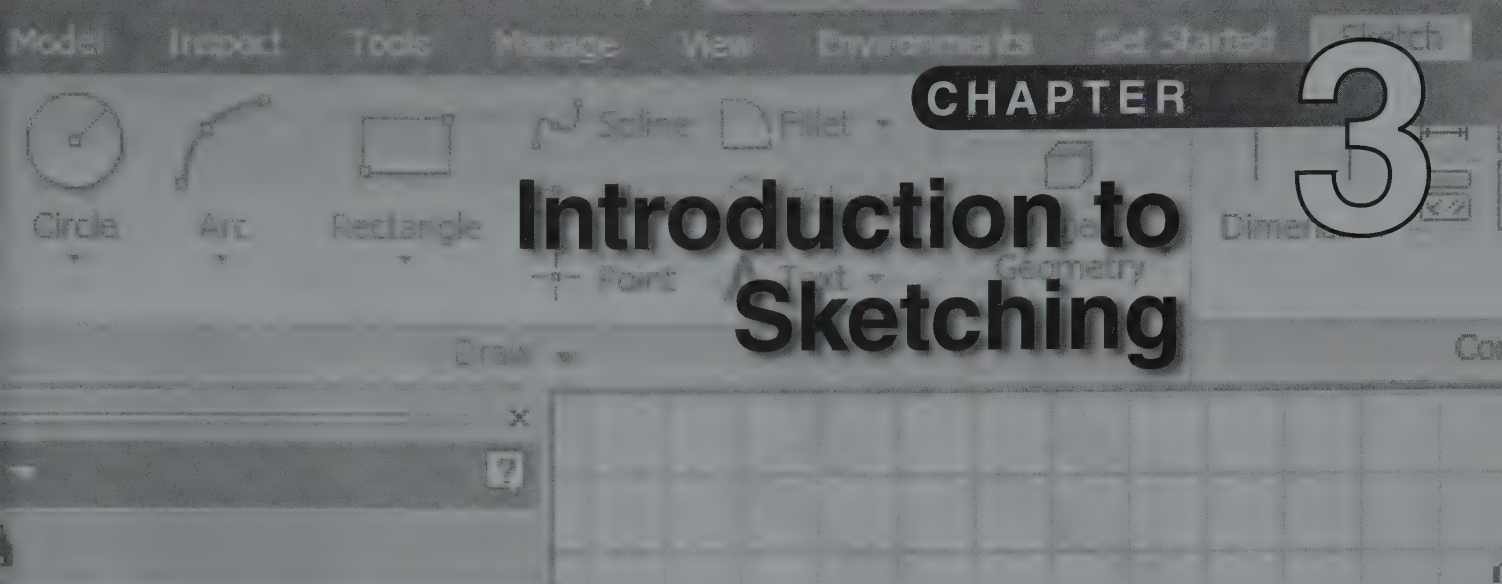
Introduction to Sketching

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Describe and apply sketching fundamentals.
✓ Sketch lines, splines, circles, ellipses, and arcs.
✓ Infer geometric constraints and set sketch linetype.
✓ Sketch rectangles and polygons.
✓ Add fillets, rounds, and chamfers to sketch geometry.
✓ Add center points and sketch points.
✓ Create sketch text.

Sketches usually represent the first step in the creation of a model. Sketches provide *profiles, paths,* and other reference geometry needed for developing sketched features. You use sketches throughout the model design process and when adding content to drawings. This chapter introduces 2D sketching and describes several 2D sketch geometry tools.

profile: The side or section outline of a sketched feature.

path: A guide, or route, for creating sketched features; directs the route of a profile.

Sketch Fundamentals

You can use several different methods or combinations of methods to create a sketch. Typically, the first step involves sketching basic shapes using sketch geometry tools, such as **Line**. See **Figure 3-1A**. Next, constrain the sketch as shown in **Figure 3-1B**. Once you finalize the shape, the sketch is ready to produce a feature. See **Figure 3-1C**.

Purpose of Sketches

The purpose of the sketch and overall design of the model influence the location, orientation, complexity, and characteristics of the sketch. For example, **Figure 3-2** shows a basic sketch used as a profile to create a base feature. A single dimensional constraint controls the diameter of the base feature. Adding several features creates the final part, a valve stem. In contrast, **Figure 3-3** shows a complex sketch used as a profile to create a base feature. Many dimensional constraints control a significant portion of the final part geometry. Adding a small number of features completes the part, a flat spring. Notice that the sketch in **Figure 3-2** is oriented horizontally, and the

Figure 3-1.
A—Initial sketch geometry is often a basic outline. Use the **Line** tool to sketch this shape.
B—Define sketches using geometric and dimensional constraints. Coincident, horizontal,
perpendicular, and parallel geometric constraints are required to constrain this sketch.
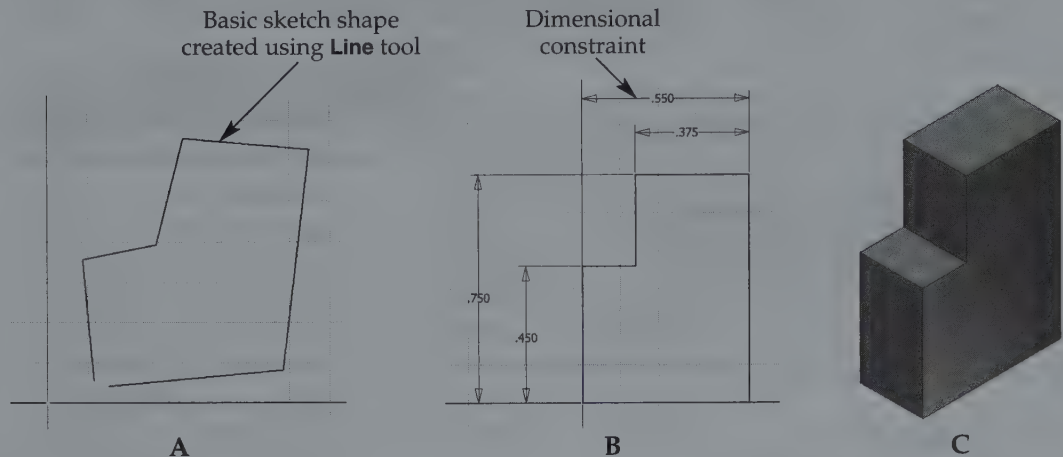C—A sketched feature built from the final sketch.

Basic sketch shape
created using **Line** tool

Dimensional
constraint

.550

.375

.750

.450

A

B

C

Figure 3-2.
A—A sketched circle defines the diameter of the base feature. B—A base feature created
by extruding the circle. C—The finished valve stem requires several additional features to
modify the base feature.

The finished
valve stem

Base
feature

Sketch for
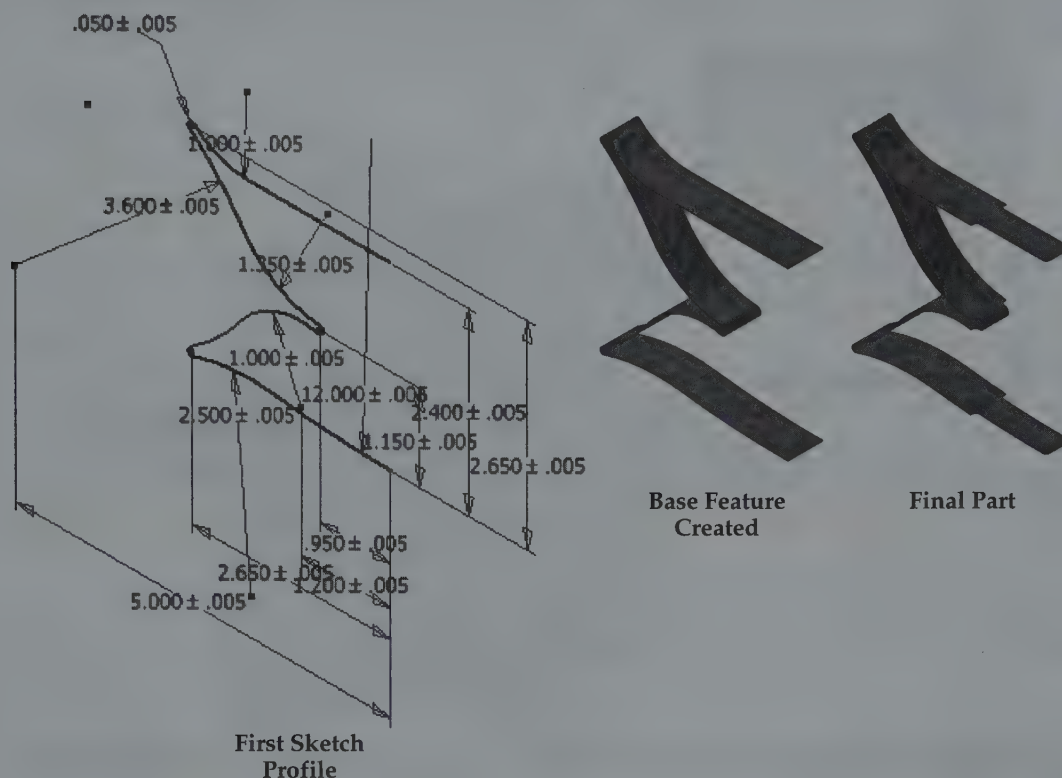base feature

.390

A

B

C
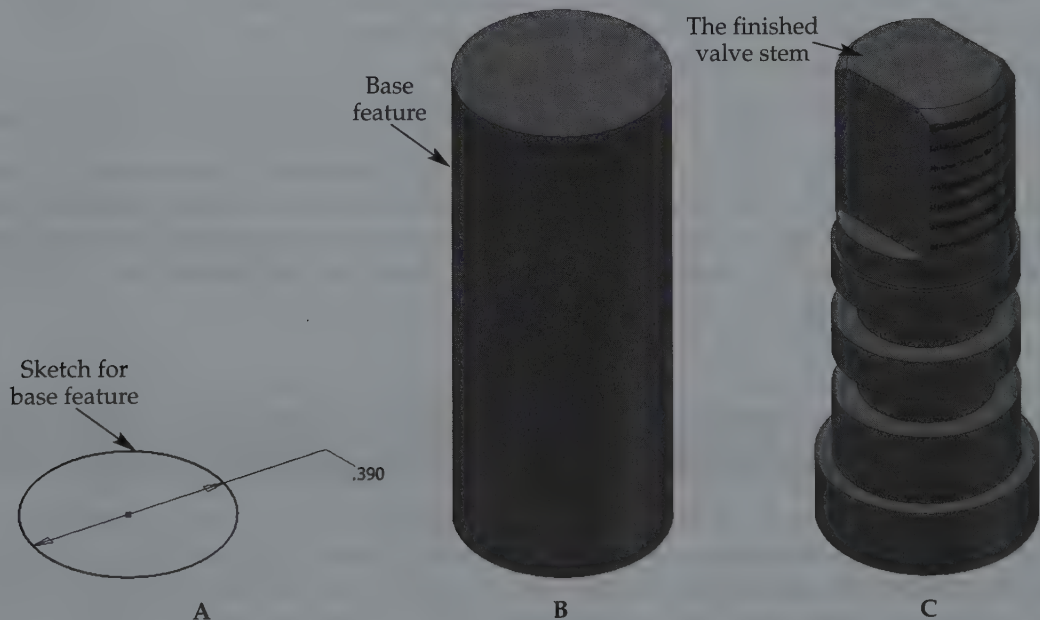
sketch in **Figure 3-3** is oriented vertically. Choose the appropriate sketch orientation
for each sketch, depending on the characteristics and orientation of the related feature,
part, and eventual drawing.

Guidelines

Sketching is one of the most critical and often time-consuming phases of model
development. Proper sketching techniques significantly increase productivity and help
ensure the creation of correct models and drawings. Consider the following guidelines
when developing sketches:

Figure 3-3.
A complex sketch used to create the base feature and the primary geometry of a flat spring.

.050 ± .005

1.000 ± .005

3.600 ± .005

1.350 ± .005

1.000 ± .005

12.000 ± .005

2.500 ± .005

2.400 ± .005

1.150 ± .005

2.650 ± .005

.950 ± .005

2.650 ± .005

1.200 ± .005

5.000 ± .005

First Sketch
Profile

Base Feature
Created

Final Part

- Initial sketch geometry is often approximate. Sketch a basic shape and then finalize the shape using constraints.
- Develop basic sketches with minimal dimensional constraints when appropriate.
- Use features to develop a model when possible. For example, add a placed chamfer feature instead of a sketched chamfer feature.
- Create a *closed loop* profile when possible. Most features created using an *open loop* are volumeless surfaces.
- Fully constrain a sketch to preserve design intent, unless the sketch shape should be able to adjust to changes made to associated geometry.

closed loop: A sketch that does not contain any gaps or openings.

open loop: A sketch that includes a gap between objects.

PROFESSIONAL TIP

You may have to dedicate a significant portion of model development to sketching. Accurate sketches that are appropriate for the model minimize design errors, increase revision effectiveness, and reduce time spent designing and drafting.

2D plane: A flat, infinite 2D surface.

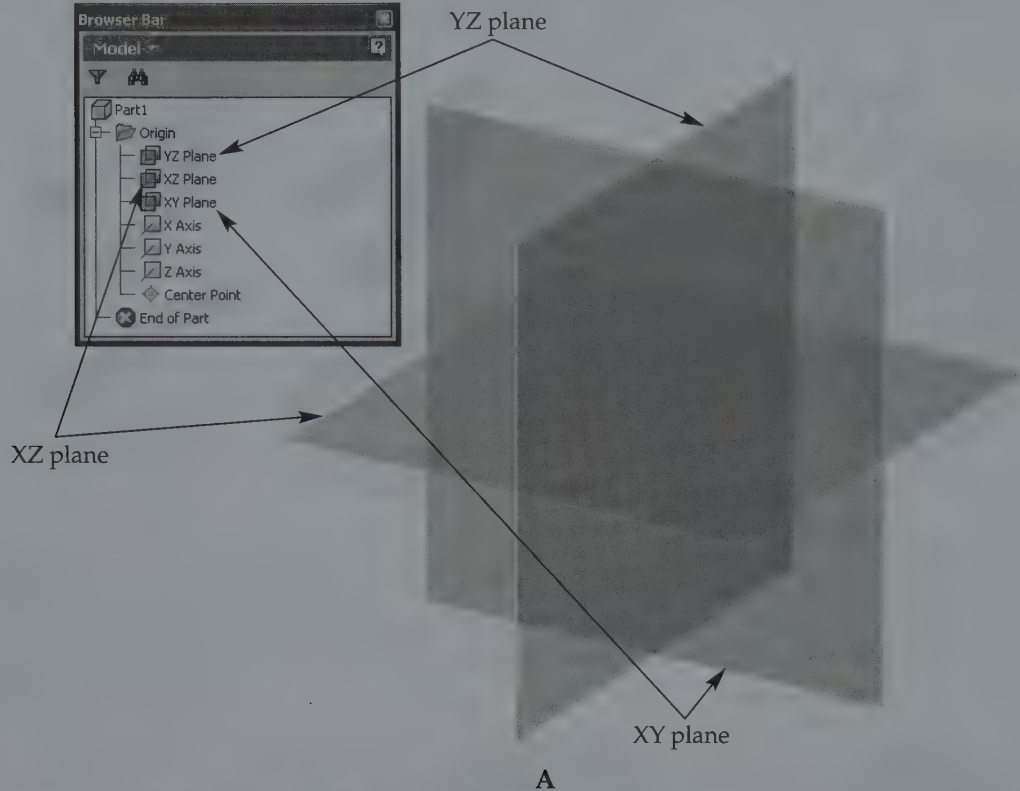**center point:** The point at the intersection of the X, Y, and Z axes in 3D space, or 0,0,0.

2D Sketching

A 2D sketch for a base feature typically occurs on one of the default *2D planes* listed in the **Origin** folder of the browser. See **Figure 3-4**. Sketches for additional features can also reference origin planes. Each plane intersects at the *center point* of the *coordinate system*. See **Figure 3-5**.

coordinate system: The system of XYZ coordinate values that defines the location of points in 3D space.

Figure 3-4.
A—Position sketches using the 2D planes available in the **Origin** folder of the browser. B—A description of each of the default planes.

YZ plane

Browser Bar
Model
Part1
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
End of Part

XZ plane

XY plane

A

Plane	Direction	View (depending on view orientation)
YZ	Vertical	Right-side or left-side
XZ	Horizontal	Top or bottom
XY	Vertical	Front or rear

B

Figure 3-5.
A—The model coordinate system uses three axes (X, Y, and Z) to define point locations in 3D space. B—Use the 3D indicator located in the lower-left corner of the graphics window to help identify the model's orientation in 3D space.
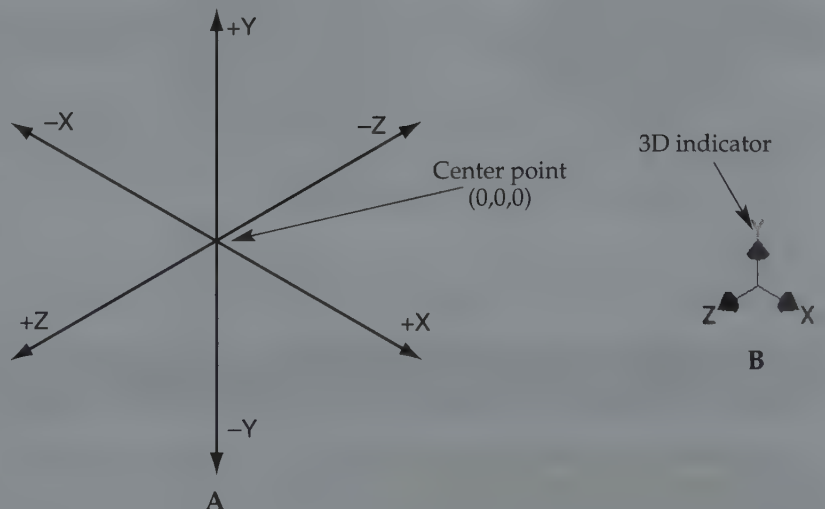
+Y

−X

−Z

3D indicator

Center point
(0,0,0)

+Z

+X

Z X

B

−Y

A

Beginning a Sketch

By default, when you start a new part model, a sketch automatically forms on the XY plane, the plane rotates perpendicular to your line of sight, and you are ready to begin sketching. You know you are in the sketch environment when you see a sketch active in the browser (Sketch1 by default) and the **Sketch** ribbon tab. See **Figure 3-6**. An active sketch appears as the only item in the browser that does not display a gray background.

The **Sketch on new part creation** area in the **Part** tab of the **Application Options** dialog box controls if and on what plane a sketch opens when you start a new part. The **Sketch on x-y plane** radio button is selected by default. Choose a radio button that corresponds to a different plane, or pick the **No new sketch** radio button. The **No new sketch** option provides the greatest flexibility for beginning a new part by allowing you to choose the plane on which to start the sketch.

Use the **2D Sketch** tool to begin a 2D sketch manually. You can access the tool first and then select a plane, or pick a plane before accessing the tool. The **2D Sketch** tool allows you to place a sketch on a plane in the **Origin** folder or on any other face or plane, as is often necessary when creating additional features.
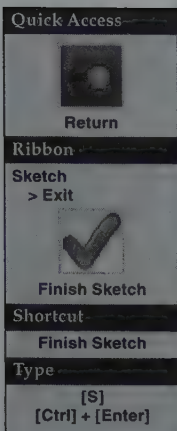
Ribbon

Model
> Sketch

Create 2D Sketch

Shortcut

New Sketch

Type

[S]

Figure 3-6.
The default Inventor sketch work environment.
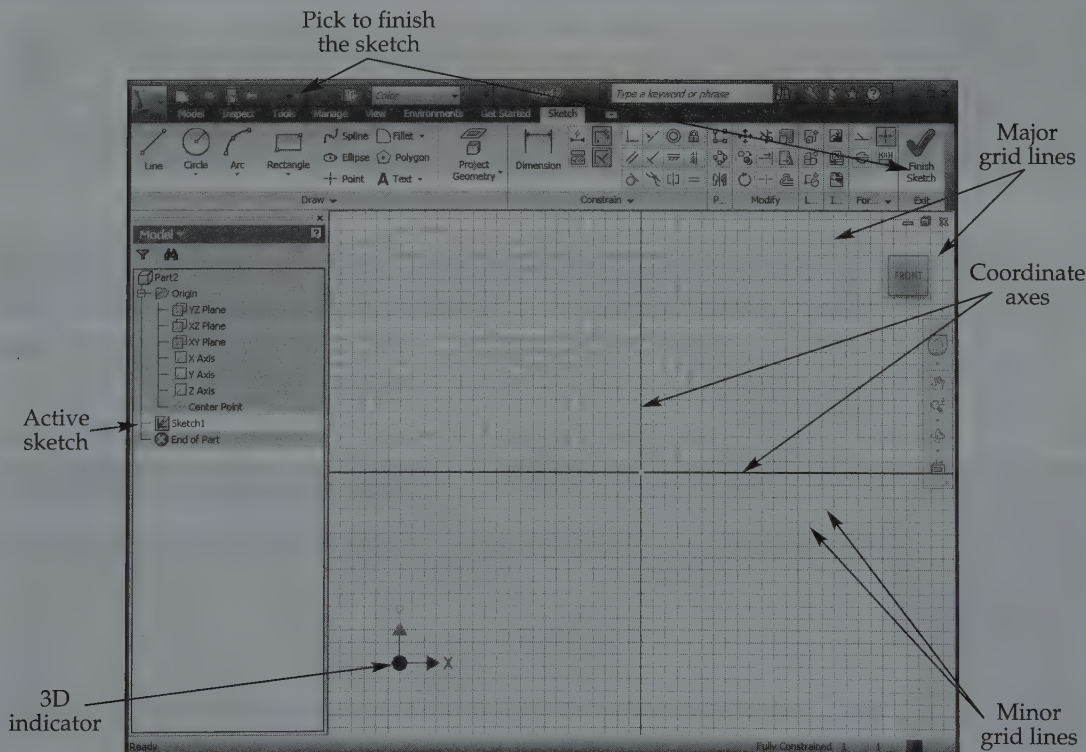
Finishing a Sketch

When you finish creating a sketch, exit sketch mode using the **Return** tool. To automate the process of finishing a sketch and initiating a feature tool while in sketch mode, right-click and select an option from the **Create Feature** cascading menu, or pick a feature tool from the **Model** ribbon tab. You know you have finished the sketch and entered the feature work environment when the **Sketch** tab and **Exit** panel disappear from the ribbon, the **Return** tool is disabled, and browser items no longer display a gray background.

Editing a Sketch

To edit a sketch, double-click on the sketch in the browser or right-click on the sketch in the browser or the graphics window and pick **Edit sketch**. Delete an unneeded sketch by picking the sketch in the browser and pressing [Delete] or right-clicking on the sketch in the browser and selecting **Delete**. If a sketch is on the incorrect plane, delete and recreate the sketch on a different plane, or redefine the sketch plane. To redefine the sketch plane, right-click on the sketch in the browser and select **Redefine**. Then pick a different sketch plane, such as a plane from the **Origin** folder. Any constraints associated with the original sketch plane, such as a constraint to the center point, are removed when you change the sketch plane.

Basic Sketch Viewing

You can prepare a 2D sketch in a 3D view orientation, but it is typically most effective to sketch while the sketch plane is perpendicular to your line of sight. When first learning Inventor, you may be in a view position that does not allow you to see the sketch plane. You can notice immediately if the view orientation is not appropriate if you do not see the default sketch grid lines, shown in **Figure 3-6**.

Use the **View Face** tool to rotate the display of the sketch plane perpendicular to your line of sight. Once you access the **View Face** tool, pick the sketch in the browser or visible sketch geometry in the graphics window.
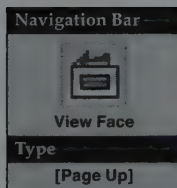
> **PROFESSIONAL TIP**
>
> Set Inventor to rotate the display perpendicular to a new sketch plane by picking the **Look At sketch plane on sketch creation** check box in **Sketch** tab of the **Application Options** dialog box. This is a convenient option, although you may feel more comfortable using view tools to adjust the display manually.

To *zoom out*, roll the mouse wheel forward. To *zoom in*, roll the mouse wheel back. To *pan*, press and hold the mouse wheel and then move the mouse, or press the arrow keys. You will discover many other view tools throughout this textbook. Some of these tools are specific to viewing 3D models, while others apply to models and sketches. For now, the **View Face** tool and basic zoom and pan functions will allow you to view sketches effectively.

> **NOTE**
>
> Toggle the display of sketch plane axes and grid lines using check boxes in the **Display** area in the **Sketch** tab of the **Application Options** dialog box. To snap to grid lines, select the **Snap to grid** check box in the **Sketch** tab. The **Sketch** tab of the **Document Settings** dialog box controls snap and grid spacing and display characteristics. Axis and grid lines are off and the graphics window background is white in this textbook for clarity.

Quick Access

Return

Ribbon
Sketch
> Exit

Finish Sketch

Shortcut
Finish Sketch

Type
[S]
[Ctrl] + [Enter]

Navigation Bar

View Face

Type
[Page Up]

zoom out: Reduce the displayed size of objects in the graphics window to view a larger area in less detail.

zoom in: Increase the displayed size of objects in the graphics window to view a smaller area in greater detail.

pan: Reposition the display of objects in the graphics window.

Referencing Origin Features

Inventor provides options for using existing geometry for sketch development. Many reference options involve projecting existing features onto the sketch plane, as described later in this textbook. For now, reference the work features available in the **Origin** folder of the browser. You must project an item onto the sketch plane in order to reference the item in the sketch. Project the center point, or *origin*, the YZ, XZ, or XY plane, or the X, Y, or Z axis before you sketch any geometry. Then use the projected point, plane, or axis to begin a sketch from the origin feature. This ensures that the sketch references a logical position in 3D space and provides appropriate geometry to define the sketch.

origin: The center point (0,0,0) of the model's XYZ coordinate system.

> **NOTE**
>
> In most cases, project the **Center Point** in the **Origin** folder to reference for sketch development.

Use the **Project Geometry** tool to project work features from the **Origin** folder and from existing feature geometry onto the sketch plane. Once you access the **Project Geometry** tool, pick the geometry or work feature to reference. To exit, press [Esc], right-click and pick **Done**, or access another tool.

Ribbon
Sketch
> Draw

Project Geometry
Shortcut
Project Geometry
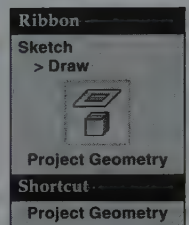
> **NOTE**
>
> You can automatically project the center point on a new sketch plane by picking the **Autoproject part origin on sketch create** check box on the **Sketch** tab in the **Application Options** dialog box. Selecting this option eliminates the need to project the center point manually to reference the origin to constrain a sketch. Be aware, however, that the center point projects onto each new sketch plane, which is often not necessary.

Exercise 3-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-1.

Ribbon
Sketch
> Draw

Line
Shortcut
Create Line
Type
[L]
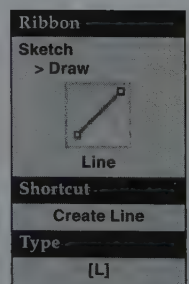
Lines

Access the **Line** tool to sketch a line. Pick the first point, followed by the next point. See **Figure 3-7**. Pick additional points to sketch connected lines. To sketch disconnected lines or restart the tool if you pick an incorrect point, press [Enter] or right-click and select **Restart**. To exit, press [Esc], right-click and pick **Done**, or access another tool.
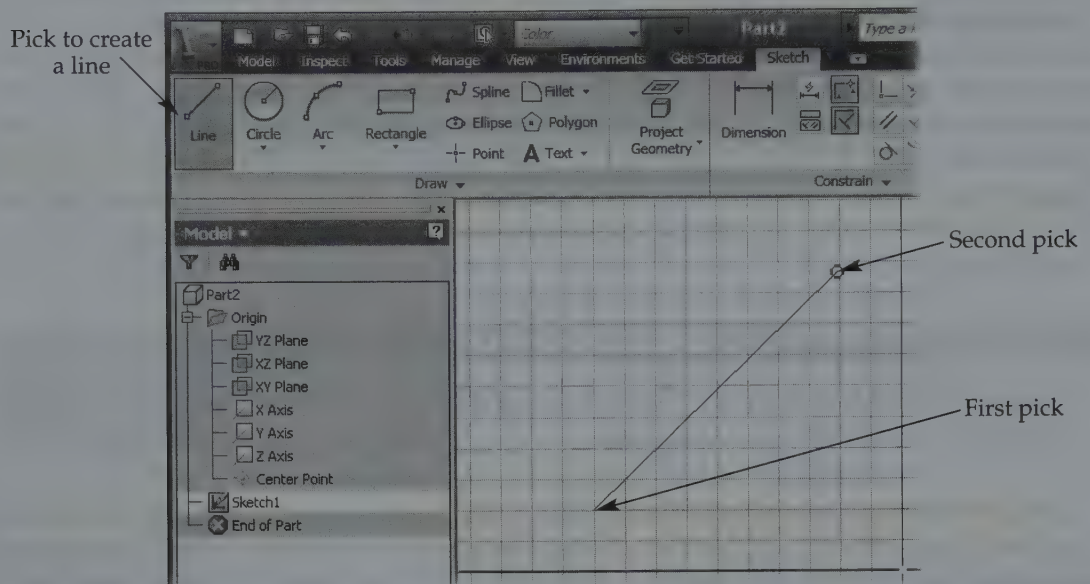
> **NOTE**
>
> Inventor considers a line to be a *curve*.

curve: A straight or bent continuous object, such as a line, arc, spline, or circle.

Figure 3-7.
To sketch a line, access the **Line** tool and pick a start point and an endpoint.

Pick to create a line

Second pick

First pick

> **PROFESSIONAL TIP**
>
> When creating sketch geometry, use the coordinates and measurements displayed near the right side of the status bar. The information helps you create an initial sketch using approximate dimensions and assists you in understanding geometric placement. However, the coordinates and measurements do not constrain a sketch. Do not spend too much time trying to use exact measurements. Add dimensional constraints later to define object size and location.

Selecting Objects

Many sketch tools and operations require you to select objects. For example, to delete a line, select the line to delete and then press [Delete] or right-click and pick **Delete**. Select an individual object by left- or right-clicking on the item, depending on the operation. If multiple objects are close together or stacked, you may be able to use the **Select Other** tool to cycle through and select the appropriate object. Hover the cursor over objects until you see the **Select Other** tool. See **Figure 3-8**. If the tool does not appear automatically, you may be able to access the tool by right-clicking and picking **Select Other…**. You can also apply these selection techniques to models to pick points, edges, and faces.

You will often need to select multiple objects at the same time. For example, to delete an octagon created using the **Polygon** tool, you must select all eight lines. Hold down [Ctrl] to select multiple objects precisely. *Window selection* or *crossing selection* is a quicker, but often less precise, selection method. To apply window selection, hold down the left mouse button above or below and to the *left* of the objects to select. Then drag the corner of the box to the right and up or down to enclose the objects to select completely, and release the mouse button. See **Figure 3-9A**.

To apply crossing selection, hold down the left mouse button above or below and to the *right* of the objects to select. Then drag the corner of the box to the left and up or down, across the objects to select. See **Figure 3-9B**. By default, the crossing selection box displays a dashed outline with a light green background to distinguish it from the window selection box, which displays a solid outline and light pink background.

window selection: Selecting objects using a window box. Only items entirely within the window are selected.

crossing selection: Selecting objects using a crossing box. Objects contained within the box and objects touching the box are selected.

Figure 3-8.
Use the **Select Other**
tool to help identify
and select close or
stacked objects.

Highlighted
selection

Pick to cycle through
and highlight possible
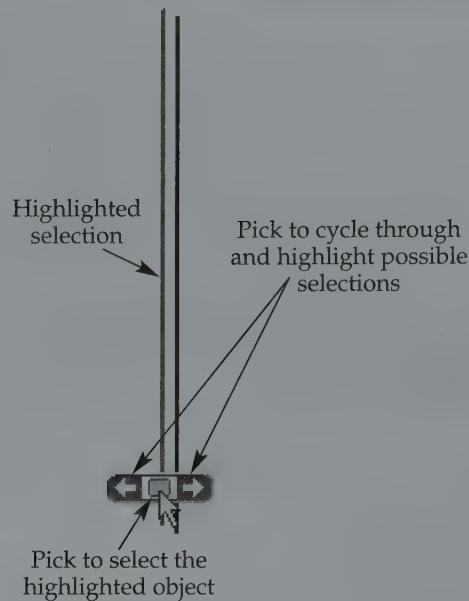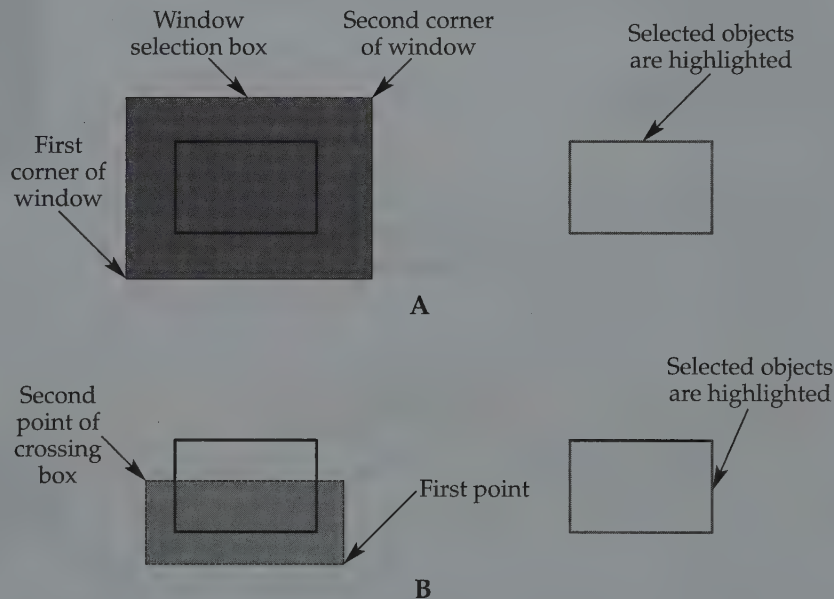selections

Pick to select the
highlighted object

Figure 3-9.
A—Use window
selection to pick all
objects completely
inside a window
selection box.
B—Use crossing
selection to pick
all objects inside
or touching the
crossing selection
box.

Window
selection box

Second corner
of window

Selected objects
are highlighted

First
corner of
window

A

Second
point of
crossing
box

Selected objects
are highlighted

First point

B

NOTE

The **Selection** area in the **General** tab of the **Application Options** dialog
box controls selection options. The **Locate tolerance** text box controls
how far away from an object you can pick and still select the object.
The distance is a number of pixels from 1 to 10. Use the **"Select Other"**
delay (sec) text box to specify the number of seconds you can hover
over model geometry before the **Select Other** tool appears.

PROFESSIONAL TIP

To select all items in the graphics window while using certain tools
that require selections, right-click and pick **Select All**.

Choosing Linetype

Sketch linetype determines the function and style of sketch geometry. The default format creates sketch geometry. Use sketch geometry to create objects recognized as a feature profile or path. See **Figure 3-10A**. It may help to consider the sketch geometry format as the visible object lines on a sketch. Sketched objects that use the sketch geometry style appear thick and solid. To apply a different linetype to sketch objects, select the appropriate button from the **Format** panel of the **Sketch** ribbon tab before sketching, or select existing objects followed by picking a button to convert the linetype.

Use *construction geometry* for geometric construction purposes. Inventor does not recognize construction geometry when you select sketch objects to create features. See **Figure 3-10B**. Therefore, you should use construction geometry for all construction purposes. Objects sketched using the construction geometry format appear thin and dashed.

construction geometry: Geometry used for construction purposes only. Inventor cannot use construction geometry to build sketched features.

Figure 3-10.
A—Objects sketched using the sketch geometry format from the feature profile.
B—Construction line format used to create reference geometry.
C—This feature recognizes the centerline format as the axis of revolution.

.680

Sketch profile created using the sketch geometry line style

Sketched feature created by selecting the sketch profile

A

.500
fx:.250
.750

Construction circle is not recognized when you select the sketch profile

Circle sketched using construction line style for construction purposes only

Sketch profile created using sketch geometry line style

B

Sketch profile
Ø.600
Ø.500
.750
2.000
Ø.400

Axis line sketched using centerline style

Revolution profile and axis automatically recognized and selected

C

Sketch *centerlines* using the centerline style. Objects sketched using the centerline style appear thick and use a centerline linetype. Inventor recognizes a sketched centerline as an axis for creating features that require an axis, such as a revolution. See **Figure 3-10C**. However, you should limit the use of centerlines to sketch requirements that truly require a centerline or line of symmetry. You can apply a construction and centerline format to sketch geometry to form construction centerlines. Use this technique to create a centerline for construction purposes that is not selectable for feature creation.

> **centerline:** A line that defines an axis of symmetry or the center of a circular feature.

Ribbon
Sketch
> Format
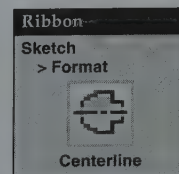
Centerline

NOTE

You can only apply a centerline format to linear objects, such as lines. While sketching a line or spline, activate or deactivate a centerline and/or construction style by right-clicking and picking the appropriate menu option. The **Construction** shortcut menu option is also available for sketching splines.

Exercise 3-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-2.

> **geometric constraints:** Geometric restrictions applied to define sketch geometry in reference to other sketch geometry.
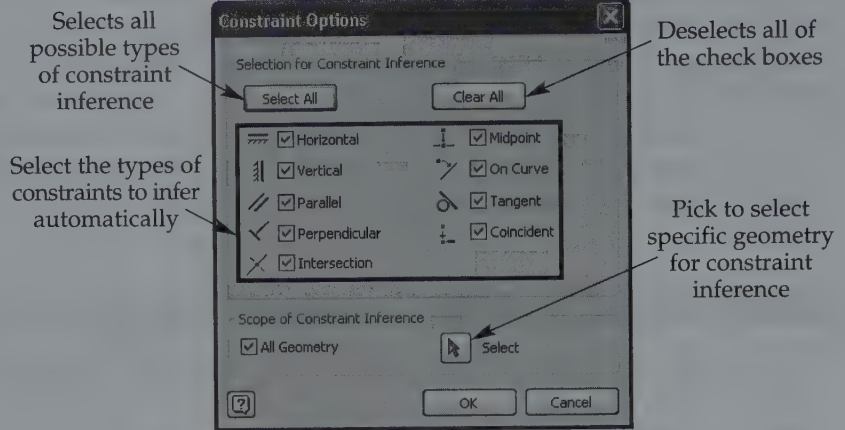
Inferring Geometric Constraints

Geometric constraints apply common geometric constructions such as two perpendicular lines, equal-sized objects, or a line tangent to a circle. Geometric constraints are required to define a sketch. You can *infer* geometric constraints while sketching or add geometric constraints after sketch geometry is in place. Inferring constraints is a very effective way to add geometric sketch relationships. Often, however, a combination of inferred and manually added geometric constraints is necessary.

The **Constraint Inference** tool controls constraint inferencing and is active by default. In order for inferred constraints to apply, the **Constraint Persistence** tool must also be active, which is the default. If you deselect the **Constraint Inference** or **Constraint Persistence** button, geometric constructions appear for reference only, and actual constraints do not form.

Access the **Constraint Options** dialog box, shown in **Figure 3-11**, and select the check boxes to specify which constraints you want to infer. You should typically infer constraints for all new and existing objects. However, to limit inferencing to specific objects, deselect the **All Geometry** check box and select the object(s) to isolate. For example, if you sketch two circles, deselect the **All Geometry** check box, and select one of the circles to include, you will be able to infer a tangent constraint only between a new line and the selected circle.

The **Constraint Placement Priority** area in the **Sketch** tab of the **Application Options** dialog box further controls constraint inferencing. Select the **Parallel and Perpendicular** radio button to infer *parallel* and *perpendicular* constraints before or instead of horizontal and vertical constraints. Pick the **Horizontal and Vertical** radio button to infer horizontal and vertical constraints before or instead of parallel and perpendicular constraints.

> **infer:** Automatically detect and apply using logic.

Ribbon
Sketch
> Constrain

Constraint Inference

Ribbon
Sketch
> Constrain

Constraint Persistence

Shortcut
> Constraint Options...
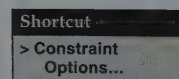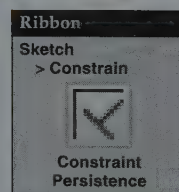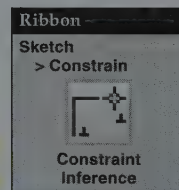
> **parallel:** A geometric construction that specifies that objects such as lines and ellipse axes will never intersect, no matter how long they become.

> **perpendicular:** A geometric construction that defines a 90° angle between objects such as lines and ellipse axes.

Figure 3-11.
Use the **Constraint Options** dialog box to specify the constraints to infer and apply to objects.

Selects all possible types of constraint inference

Deselects all of the check boxes

Select the types of constraints to infer automatically

Pick to select specific geometry for constraint inference

NOTE

The following information assumes that the **Line** tool is active. However, you can infer constraints when using many different sketch tools; the process is very similar.

Coincident

coincident:
A geometric construction that defines two points sharing the same location.

A *coincident* constraint is one of the most common geometric constructions and is often required to create a properly closed profile. For example, the endpoints of two lines that form a rectangle corner coincide. To infer a coincident constraint, move the crosshairs and yellow dot near existing geometry, such as the projected center point. When the cursor snaps to a position and the coincident symbol appears, pick the point to form a coincident constraint. When you reference a specific point, the yellow dot turns green and gets larger. See **Figure 3-12**.

Use coincident constraint inference to snap accurately to a specific point, such as the projected center point or a point on an existing object. Each variation creates a coincident constraint. For example, a coincident constraint formed at the midpoint of an existing line links two lines together, while a midpoint reference defines the exact location at which the lines meet.

PROFESSIONAL TIP

You can isolate the midpoint, center point, or intersection of sketch objects while sketching by right-clicking and picking **Midpoint**, **Center**, or **Intersection**. This allows you to add a coincident constraint by picking a specific existing point on an object when it is difficult to infer the exact location because of conflicting geometry. For example, pick the start point of a line, right-click and select **Midpoint**, and then pick an existing line to end the new line at the midpoint of the existing line.

Horizontal and Vertical

A horizontal constraint aligns geometry with the X axis on the XY plane, the Y axis on the YZ plane, or the X axis on the XZ plane. A vertical constraint aligns geometry with the Y axis on the XY plane, the Z axis on the YZ plane, or the Z axis on the XZ plane. To infer a horizontal or vertical constraint, move the crosshairs and yellow dot to a position near horizontal or vertical. When the cursor snaps to a position and the appropriate symbol appears, pick the point to form a horizontal or vertical constraint. See **Figure 3-13**.

Figure 3-12.
Examples of inferring
common coincident
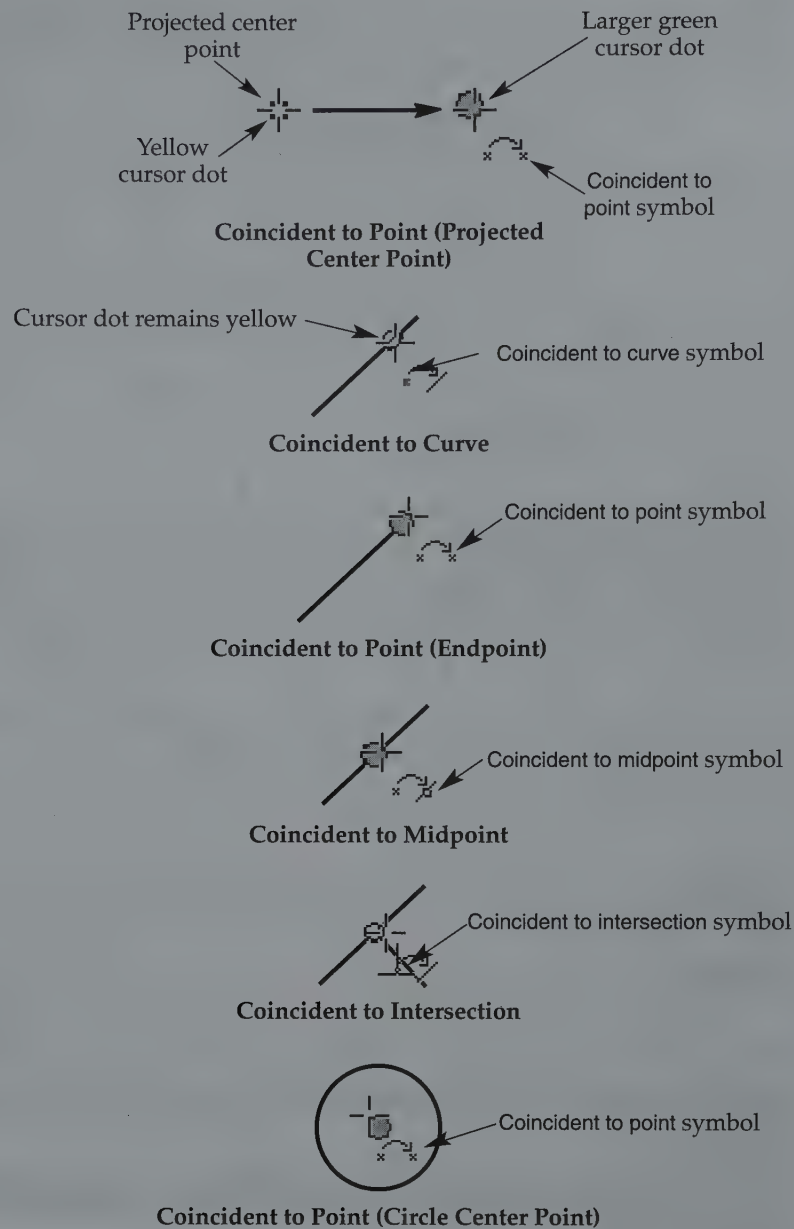constraints relative to
a specific point.

Projected center
point

Larger green
cursor dot

Yellow
cursor dot

Coincident to
point symbol

**Coincident to Point (Projected
Center Point)**

Cursor dot remains yellow

Coincident to curve symbol

Coincident to Curve

Coincident to point symbol

Coincident to Point (Endpoint)

Coincident to midpoint symbol

Coincident to Midpoint

Coincident to intersection symbol

Coincident to Intersection

Coincident to point symbol

Coincident to Point (Circle Center Point)

Figure 3-13.
Inferring horizontal
and vertical
constraints.

Pick to sketch a
vertical line

Pick to sketch a
horizontal line

Parallel and Perpendicular

To infer a parallel or perpendicular constraint, move the crosshairs and yellow
dot near an existing line, projected default work plane, or work axis. When the cursor
snaps to a position and the correct symbol appears, pick the point to form a parallel
or perpendicular constraint. See **Figure 3-14A**. You can also infer a perpendicular
constraint by holding down the left mouse button on a line to establish a coincident

Figure 3-14.
A—Inferring parallel and perpendicular constraints. B—An alternative method of inferring a perpendicular constraint from an existing object.

Symbol identifies the object
perpendicular to the new line

Symbol identifies the object
parallel to the new line

Pick to sketch a
perpendicular line

Pick to sketch a
parallel line

A

Pick and hold to establish
a coincident constraint

Release at the
appropriate location

B
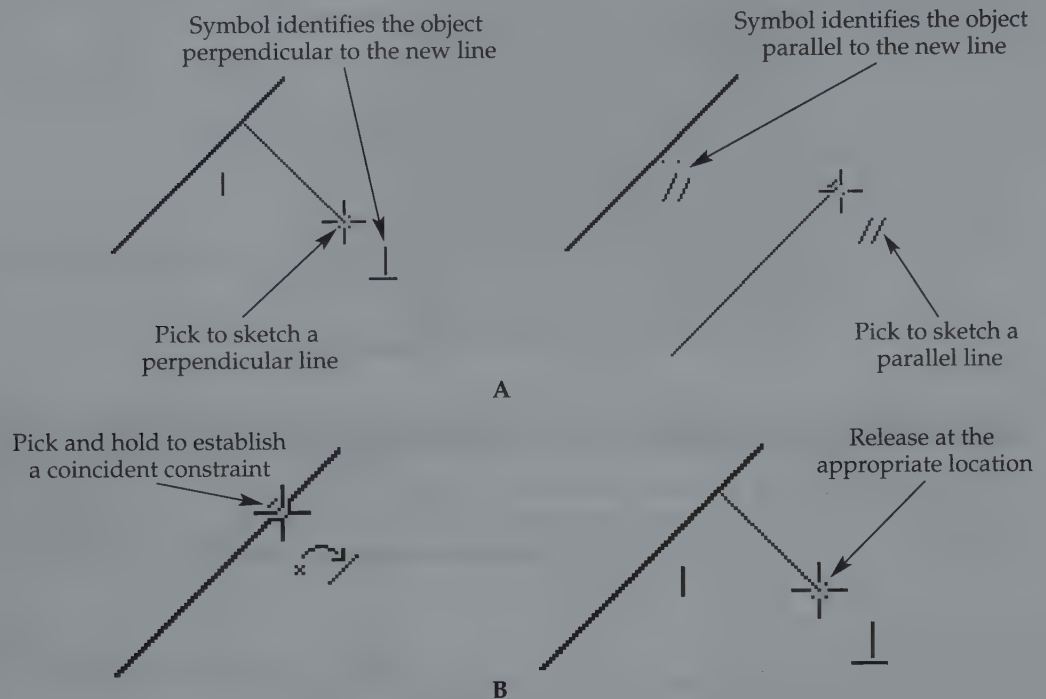
constraint, and while still holding the button down, moving the cursor away from the line. See **Figure 3-14B**.

Tangent

tangent: A geometric construction that specifies how a curve touches another curve at the point of tangency.

To infer a *tangent* constraint, move the cursor over or near an existing circular curve. When you are close to tangent, the cursor snaps to the point of tangency and the tangent symbol appears. Pick the point and continue sketching the line. See **Figure 3-15**.

> **NOTE**
>
> Some sketching tools automatically apply certain constraints to a sketch. For example, the **Rectangle** tool creates a rectangle with parallel, perpendicular, coincident, and horizontal constraints to make a true rectangle.

> **PROFESSIONAL TIP**
>
> Often, multiple constraints are inferred at the same time, such as the tangent and horizontal constraints and the tangent and coincident constraints shown in **Figure 3-15**. Use caution when inferring constraints to specify appropriate geometric constructions.

Figure 3-15.
Inferring a tangent
constraint from
and to an existing
circle. Notice the
compound tangent/
horizontal and
tangent/coincident
constraints inferring
at the same time.

Pick to end a sketched
line tangent to the curve

Tangent To

Pick to sketch a
line from and tangent
to the curve

Tangent From
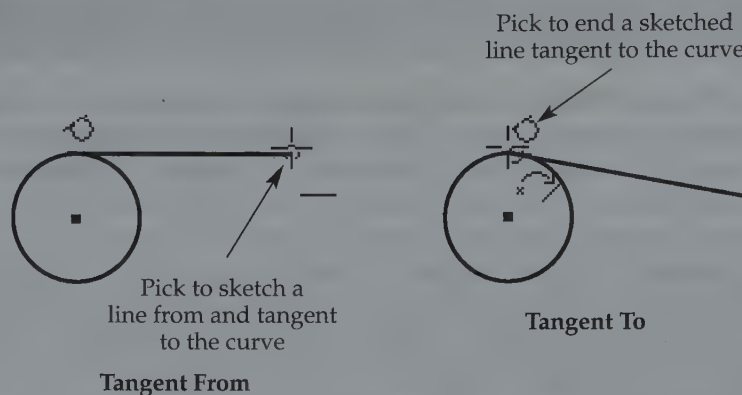
Point Alignment

Point alignment provides a way to reference a point horizontally or vertically in space with an existing sketch point, such as the endpoint or midpoint of a line or the center of a circle. Point alignment aids in referencing points to sketch accurate geometry, but it does not actually produce constraints. The **Point Alignment** check box in the **Sketch** tab of the **Application Options** dialog box is selected by default, allowing for point alignment.

When point alignment is on and you move the cursor horizontal or vertical to a point on an existing object, the cursor snaps to the appropriate position and a dotted trail appears between the cursor and the existing point. Pick the location to establish the start point or endpoint of the line. You can also reference two points, as shown in **Figure 3-16**.

If you deselect the **Point Alignment** check box, you can still use point alignment for reference, but you must first move the cursor over or near the existing object points to reference. Then move the cursor away from the object to display alignment paths. When you are far enough away from the existing geometry, and with the dotted lines still visible, pick the point.

PROFESSIONAL TIP

To deactivate most constraint inferencing temporarily, hold down [Ctrl] while sketching. Endpoint coincident constraints still are inferred.

Exercise 3-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-3.

Figure 3-16.
Use the point
alignment lines to
align new geometry
to points on existing
geometry.

Pick point

Horizontal and vertical
alignment paths

Ribbon
Sketch
 > Draw
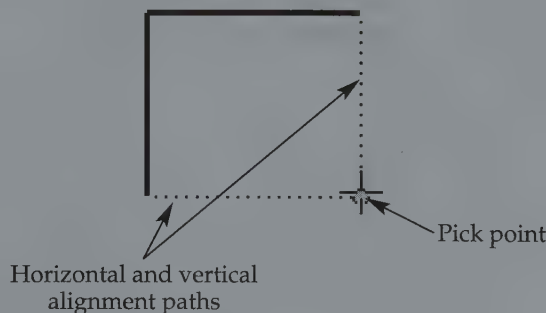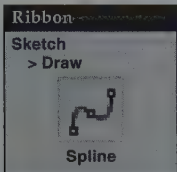
Spline

Access the **Spline** tool to sketch a *spline* object. Pick the first control point, followed by additional control points. Use constraint inferencing to form appropriate constraints while using the **Spline** tool. If you select an incorrect point, right-click and pick **Back** or press [Backspace]. To restart the tool if the first point is incorrect, right-click and select **Restart**. To end the spline, double-click the last point, press [Enter], or right-click and pick **Create**. See **Figure 3-17A**. To create a tangent closed-loop spline, move the cursor toward the spline start point. When the cursor snaps to the start point and the closed loop forms, pick the point to create the spline. See **Figure 3-17B**. Continue sketching splines, or press [Esc], right-click and pick **Done**, or access another tool to exit. Spline control functions appear, allowing you to adjust the spline as needed.

Supplemental Material

Spline Options
For information about controlling the shape of a spline, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Spline Options**.

Exercise 3-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-4.

Figure 3-17.
A—A spline is a single object with smooth, rounded corners. B—End a spline at the start point to create a tangent, closed-loop spline.
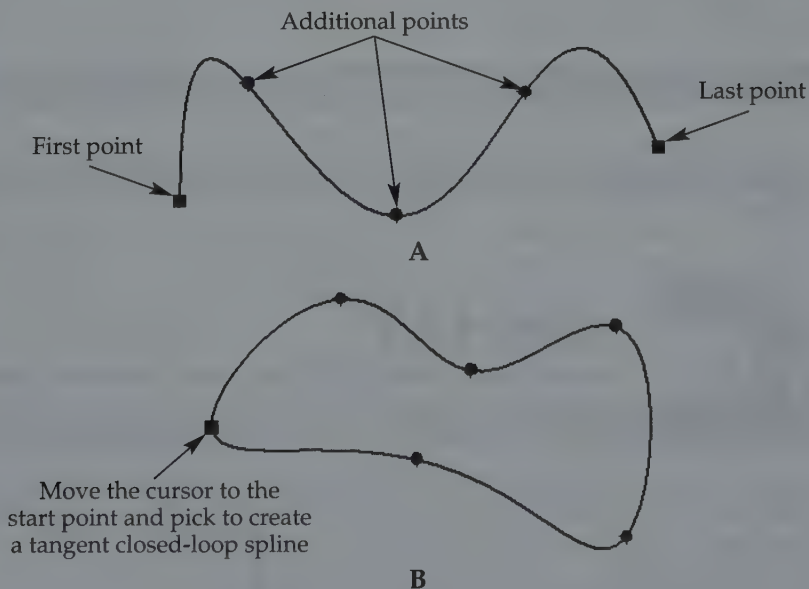
Circles and Ellipses

The **Circle** tool provides methods for sketching *circles*. Choose the appropriate option according to the information you know about constructing the circle. The **Ellipse** tool allows you to sketch an *ellipse*.

Center Point Circle

Access the **Center Point Circle** tool to sketch a circle based on the center point and radius. Pick the center of the circle, followed by the edge. See **Figure 3-18**. Continue sketching center point circles, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Tangent Circle

To sketch a tangent circle, three curves that will be tangent to the circle must be available for selection. Access the **Tangent Circle** tool and pick three existing curves. A circle forms tangent to the objects, as shown in **Figure 3-19**. Continue sketching tangent circles, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Ellipse

Access the **Ellipse** tool to sketch an ellipse. Pick the center point, followed by the first axis endpoint and then the second axis endpoint. The furthest point from the center defines the *major axis*, while the closer point defines the *minor axis*. See **Figure 3-20**. Continue sketching ellipses, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Must be placed in order
Center Major ~~first~~ minor

circle: A closed curve with a constant radius around a center point; size is usually dimensioned according to the diameter.

ellipse: An oval-like shape that contains both a major axis and a minor axis.

Ribbon
Sketch
> Draw

Center Point Circle

Type
[C]

Ribbon
Sketch
> Draw

Tangent Circle

Ribbon
Sketch
> Draw
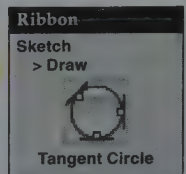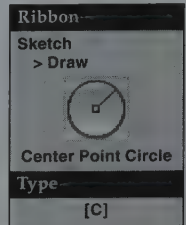
Ellipse

major axis: The longer of the two axes in an ellipse.

minor axis: The shorter of the two axes in an ellipse.

Figure 3-18.
When you create a circle using the **Center Point Circle** tool, the first pick locates the center and the second pick defines the radius.

First point (center)

Second point (radius)

Figure 3-19.
Use the **Tangent Circle** tool to sketch a circle tangent to three existing curves.

Select three objects

Tangent points

Figure 3-20.
Sketch an ellipse by picking the center point, major axis, and minor axis.
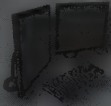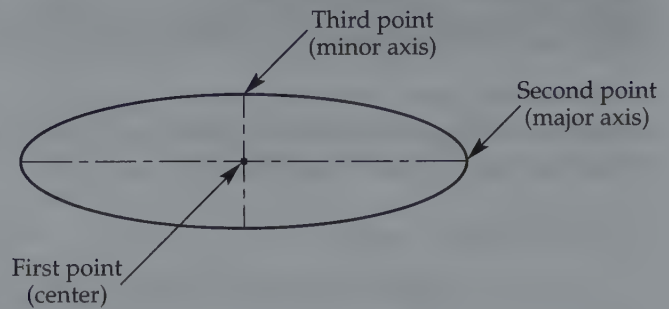
Third point (minor axis)

Second point (major axis)

First point (center)

Use constraint inferencing to form appropriate constraints while using circle and ellipse tools. Accurately select the center point, point of tangency, and outside radius or endpoint of radii and axes.
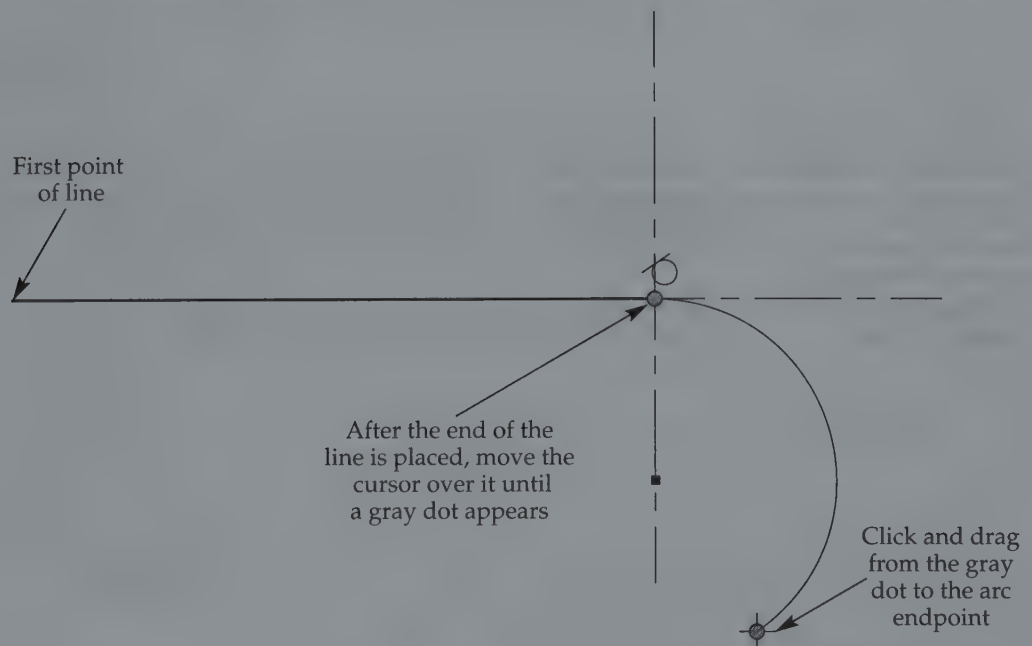
Exercise 3-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-5.

Arcs

arc: A circular curve in which all of the points are an equal distance from the center point.

The **Arc** tool provides methods for sketching *arcs*. Choose the appropriate option according to the information you know about constructing the arc. The **Line** tool also includes the ability to sketch an arc from a previously drawn line or arc.

Three-Point Arc

Ribbon

Sketch > Draw

Three Point Arc

chord length: The linear distance between two points on a circle or arc.

Access the **Three Point Arc** tool to sketch a three-point arc. Pick the start point, followed by the endpoint and then the edge. See **Figure 3-21**. The distance between the first and second points is the *chord length*. Continue sketching three-point arcs, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Figure 3-21.
When you use the **Three Point Arc** tool, the first and second points define the ends and the third point defines the size and direction of the arc.

Third point (radius)

Second point

First point

The center point appears after the arc is drawn
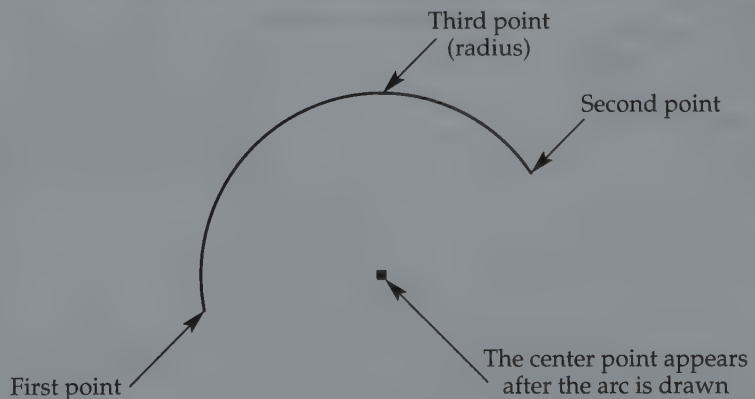
Center Point Arc

Access the **Center Point Arc** tool to sketch a center point arc by constructing the *included angle*. Pick the center point, followed by the start point and then the endpoint. See **Figure 3-22**. Continue sketching center point arcs, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Tangent Arc

To sketch a tangent arc, a curve that will be tangent to the arc must be available for selection. Access the **Tangent Arc** tool and pick an existing curve. An arc begins tangent to the selection. Pick to end the arc. See **Figure 3-23**. Continue sketching tangent arcs, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Ribbon
Sketch
> Draw

Center Point Arc

included angle:
The angle formed between the center, start point, and endpoint of an arc.

Ribbon
Sketch
> Draw

Tangent Arc

Exercise 3-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-6.

Figure 3-22.
Sketching an arc using the **Center Point Arc** tool.

First point (center)   Second point (radius)   Third point (endpoint)

The completed arc
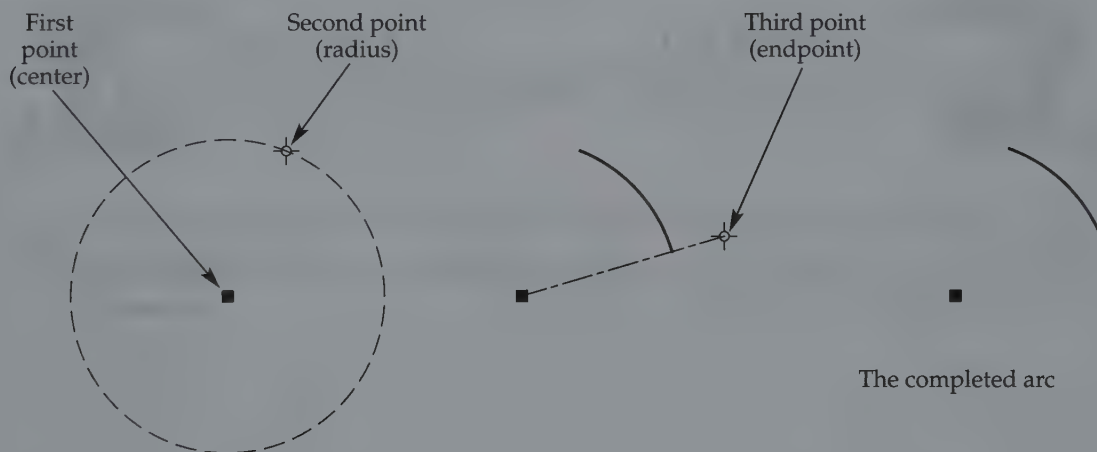
Figure 3-23.
Use the **Tangent Arc** tool to begin an arc tangent to an existing curve.

Existing curve (line)   Pick the line near the end

Pick to place the end of the arc
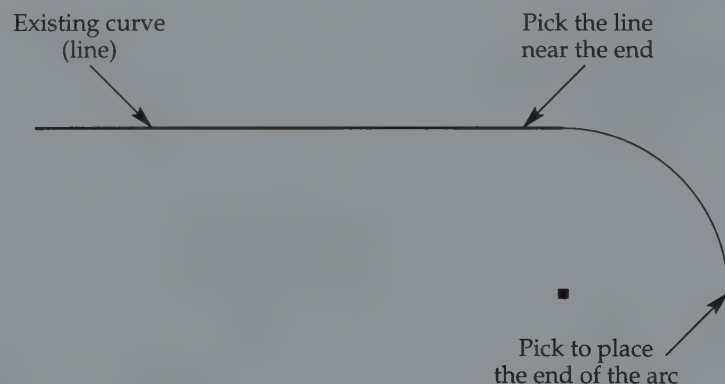
Sketching Arcs Using the Line Tool

The **Line** tool offers the ability to sketch an arc tangent to the previous line. This provides a quick way to sketch a slot profile or similar shape. Access the **Line** tool and sketch one or more lines as needed, but do not cancel or reset the **Line** tool. Move the cursor over the last specified point until the dot turns gray. Then hold down the left mouse button and move the arc endpoint to the desired location. See **Figure 3-24**. If the arc occurs in the wrong direction, without releasing the mouse button, move back to the endpoint and redefine the arc. When you are satisfied with the arc endpoint, release the mouse button.

Repeat the steps to sketch another arc, or continue to sketch a straight line from the arc. To sketch disconnected lines or restart the tool if you pick an incorrect point, right-click and select **Restart**. To exit, press [Esc], right-click and pick **Done**, or access another tool.

PROFESSIONAL TIP

Use constraint inferencing to form appropriate constraints while using arc construction tools. Accurately select the center point, point of tangency, and outside radius or radii endpoints.

Exercise 3-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-7.

Figure 3-24.
Using the **Line** tool to form a tangent arc. This technique is effective for constructing slot profiles and similar sketch geometry.

First point
of line

After the end of the
line is placed, move the
cursor over it until
a gray dot appears

Click and drag
from the gray
dot to the arc
endpoint

You can create a rectangle using the **Line** tool. However, rectangle tools are quicker to use than drawing individual line segments, and they automatically assign the appropriate coincident, parallel, perpendicular constraints. The rectangle tools produce four sides that you can delete individually.

Two-Point Rectangle

Access the **Two Point Rectangle** tool to sketch a rectangle with horizontal and vertical sides. Pick the first corner, followed by the opposite, diagonal corner. See **Figure 3-25A**. A horizontal constraint applies to the top or bottom side, depending on the location of the second point from the first. Continue sketching two-point rectangles, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Ribbon
Sketch
> Draw

Two Point
Rectangle
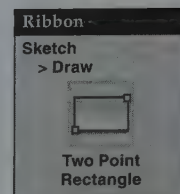
Three-Point Rectangle

Access the **Three Point Rectangle** tool to sketch a rectangle at any angle. Pick the first corner, followed by the second corner and then the third corner. See **Figure 3-25B**. Continue sketching three-point rectangles, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Ribbon
Sketch
> Draw

Three Point
Rectangle
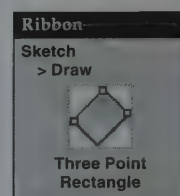
Exercise 3-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-8.

Figure 3-25.
A—Use the **Two Point Rectangle** tool to sketch a horizontally constrained rectangle. B—Use the **Three Point Rectangle** tool to sketch a rectangle at an angle.
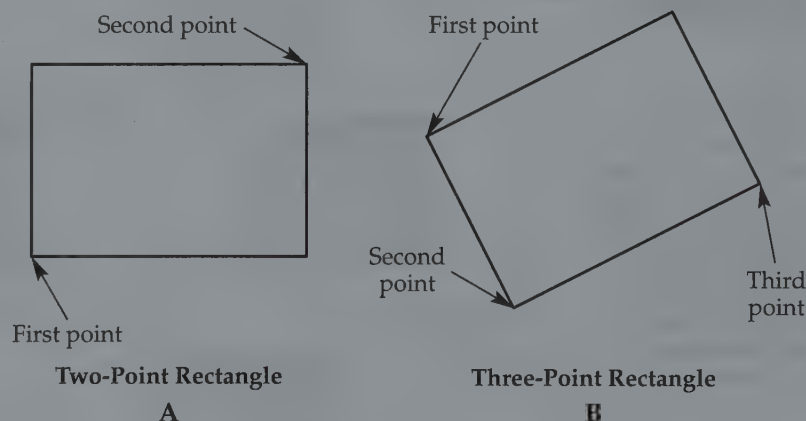
Second point

First point

Second point

Third point

First point

Two-Point Rectangle
A

Three-Point Rectangle
B

Polygons

Access the **Polygon** tool to sketch a *regular polygon* using the **Polygon** dialog box. See **Figure 3-26**. Specify whether to create an *inscribed* or *circumscribed* polygon by selecting the **Inscribed** or **Circumscribed** button. A circumscribed polygon is most common. Next, define the polygon type by entering the number of sides in the **Number of Sides** text box. Pick the center point, followed by a point to position and size the polygon. Continue sketching polygons, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

NOTE

The **Polygon** tool creates a circular pattern of edges, with pattern constraints and coincident constraints that link corners. The result is a polygon that acts as a single object. Chapter 4 explains sketch patterns.

PROFESSIONAL TIP

Use constraint inferencing to form appropriate constraints while using rectangle and polygon tools. Accurately select the center point and the corner or flat location.

Exercise 3-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-9.

Figure 3-26.
The number in the text box determines the number of sides in the polygon. The circles illustrate the difference between inscribed and circumscribed polygons.

Second point (corner)

First point (center)

Second point (flat)

Inscribed

Circumscribed
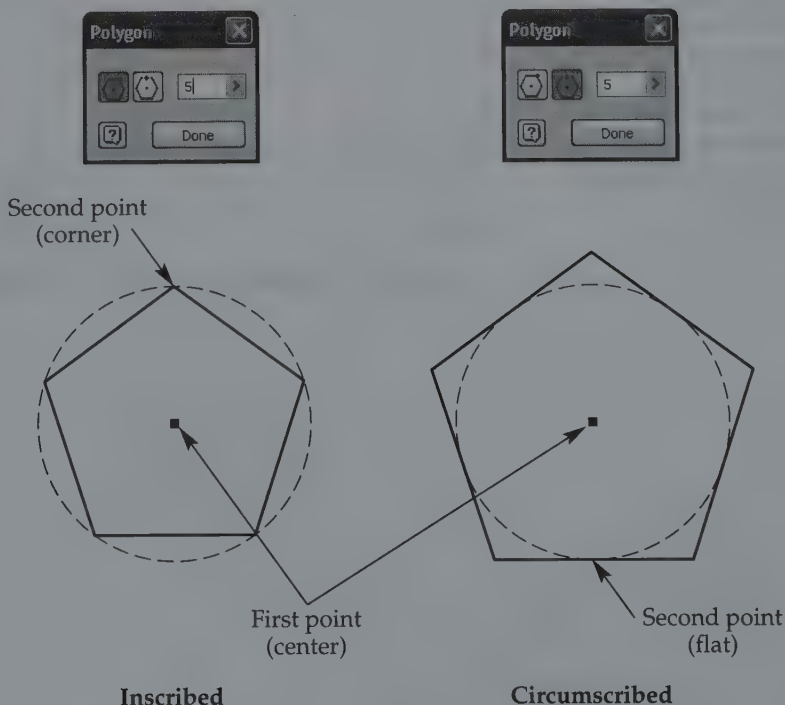
You can use sketch fillets, rounds, and chamfers, as described in this section, to create sketch geometry that controls feature size and shape. However, for many applications, you will find that placed fillet, round, and chamfer features are more appropriate. Placed features often provide more options, replicate manufacturing processes, and can be easier to create and edit.

Fillets and Rounds

Access the **Fillet** tool to sketch *fillets* and *rounds* using the **2D Fillet** dialog box. See **Figure 3-27**. Enter the fillet and round radius in the text box or pick a value from the list. Select the **Equal** button to create multiple fillets with equal radii, as shown in **Figure 3-27A**. Deselect the **Equal** button to specify dimensions for each fillet and round separately, as shown in **Figure 3-27B**.

If intersecting lines form a sharp corner, pick the corner to add the fillet or round, as shown in **Figure 3-27**. If the intersection is not a sharp corner, or if you have difficulty selecting the corner, pick individual lines as shown in **Figure 3-28**. Continue adding fillets and rounds, or close the dialog box, press [Esc], right-click and pick **Done**, or access another tool to exit.

Adding

fillet: A curve placed at the inside intersection of two or more faces, adding material to a feature.

Removing

round: A curve placed on the exterior intersection of two or more faces or corners, removing material from a feature.

Ribbon

Sketch
> Draw

Fillet

Type

[F]
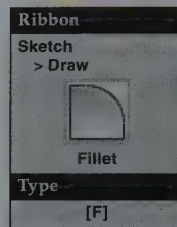
Figure 3-27.
When using the **Fillet** tool, pick two lines or the intersection of two lines to add a fillet or round. A—All fillets are controlled by one dimension if **Equal** is on. B—Each fillet is controlled individually if **Equal** is off.

Changing this dimension changes the size of all fillets and rounds

.125

Place the cursor on a corner to see a preview

2D Fillet

0.125 in

Equal button is on

A

.125

.125

Round

Fillet

Each fillet and round is controlled by its own dimension

.125

.125

2D Fillet

0.125 in

Equal button is off

.125

B
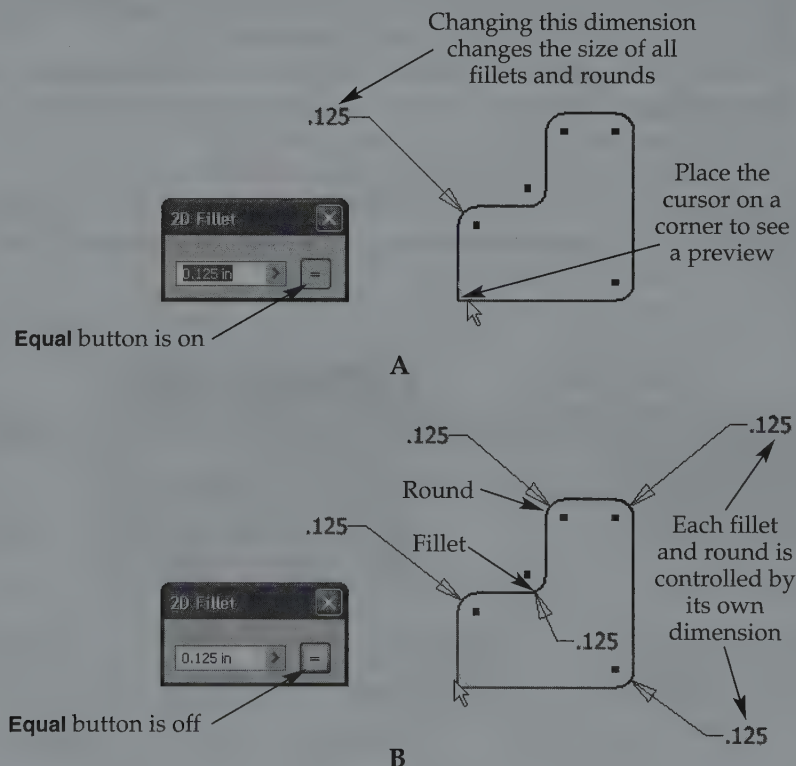
Figure 3-28.
If you select two lines when using the **Fillet** tool, Inventor can trim lines that extend past their intersection or extend lines that do not meet to form the round or fillet.

Pick the lines on the side you want to keep

—.031

Pick the individual lines to form the round

—.031
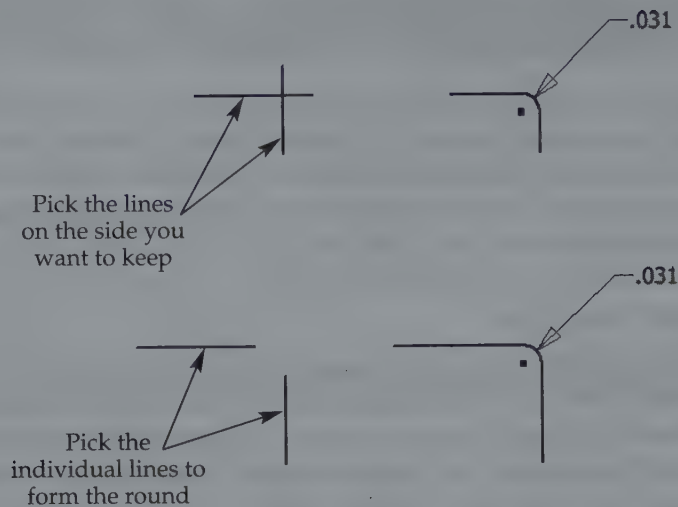
Chamfers

chamfers: Angled planar faces added to corners.

Ribbon
Sketch
> Draw

Chamfer

Access the **Chamfer** tool to sketch *chamfers* using the **2D Chamfer** dialog box. See **Figure 3-29**. Pick the **Create Dimensions** button to add dimensional constraints to the chamfer. Select the **Equal** button to create multiple, equally sized chamfers, such as when chamfering rectangle corners. Pick a chamfer type button and then enter values in the appropriate text boxes. **Figure 3-29** illustrates the process of assigning each chamfer type and using the **Create Dimensions** button.

Once you specify all of the options in the **2D Chamfer** dialog box, pick a sharp corner to add a chamfer. If the intersection is not a sharp corner, or if you have

Figure 3-29.
The options in the **2D Chamfer** dialog box control the dimensioning of chamfers.

Equal Distance button

Settings change according to chamfer type

Create Dimensions button

2D Chamfer

Distance
0.125 in

Unequal Distance button

Equal button

Distance and Angle button

OK

Fx: dimension is always equal to the other dimension

.25

fx:.25

Change one distance without affecting the other

.25

.38

Angle from the second selected line

Distance between the intersection and first selected line

.25

60.00

Equal Distance

Unequal Distance

Distance and Angle

Create Dimension deselected

difficulty selecting the corner, pick individual lines. For unequal-distance chamfers, the first selected line corresponds to the **Distance2** value and the second selected line corresponds to the **Distance1** value. For distance and angle chamfers, the first selected line corresponds to the **Angle** value and the second selected line corresponds to the **Distance** value. Pick the **OK** button to create the chamfers.

NOTE

You can also create a fillet, round, or chamfer by selecting two lines and then accessing the **Fillet** or **Chamfer** tool.

Exercise 3-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-10.

Center and Sketch Points

The **Point, Center Point** tool allows you to add *sketched center points* or *sketch points*, depending on the sketch requirement. The default is to create a center point. You often add center points to a sketch on a feature face to direct the placement of a new feature. To change from a center point style to a sketch point style, deselect the **Center Point** button in the **Format** panel of the **Sketch** ribbon tab. Select or deselect the button as needed before sketching points, or select existing points and then pick the button to convert the points to the alternate format. To use the **Point, Center Point** tool, pick to locate the point. See **Figure 3-30**. Continue sketching center and sketch points, or press [Esc], right-click and pick **Done**, or access another tool to exit.

sketched center points: Points used to define the location of center points for features that reference center points, such as holes and sheet metal punches.

sketch points: Points used for construction purposes to help develop sketch geometry.

Ribbon
Sketch
> Draw

Point, Center Point

PROFESSIONAL TIP

Use constraint inferencing to form appropriate constraints while placing center and sketch points.

Ribbon
Sketch
> Format

Center Point

Figure 3-30.
Place center points to locate features and sketch points to help construct sketch geometry.
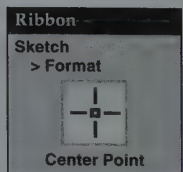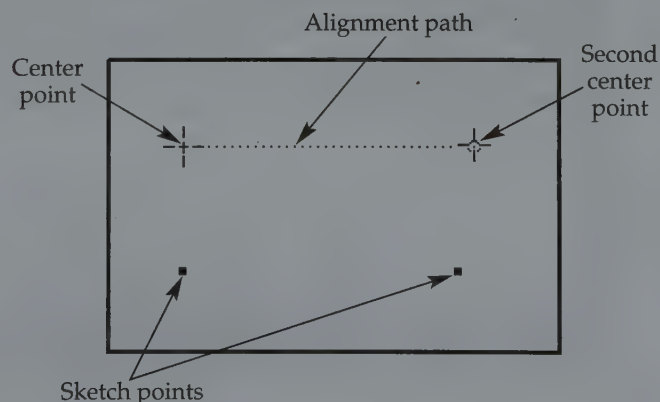
Exercise 3-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-11.

Sketch Text

In models, sketch text can be used to create features such as embossments and extrusions. In drawings, sketch text is used in title blocks, revision blocks, general notes, and other annotations. Some text settings and format options are available only when you are working in the drawing environment. The following information focuses on adding sketch text to a model.

Using the Text Tool

Ribbon

Sketch
> Draw

A

Text

Type

[T]

Access the **Text** tool to add a basic block of text. Select a point, or create a text box by holding down the left mouse button, dragging the cursor, and releasing the mouse button. The point you select, or the box you create, defines the text location and extents. The **Format Text** dialog box appears, as shown in **Figure 3-31**. Format text and insert content using the functions in the upper portion of the dialog box. Type text and apply associated operations such as copying and pasting in the large text box. When you are
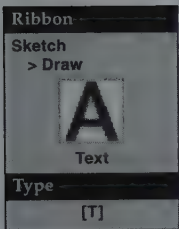
Figure 3-31.
The **Format Text** dialog box appears when you pick a location for new text or edit existing text.

Vertical justification buttons   **Text Box** button   Pick to create single line only text

Horizontal justification buttons

Color button

Pick to select and insert a symbol

Text box

Add Parameter button

Zoom buttons
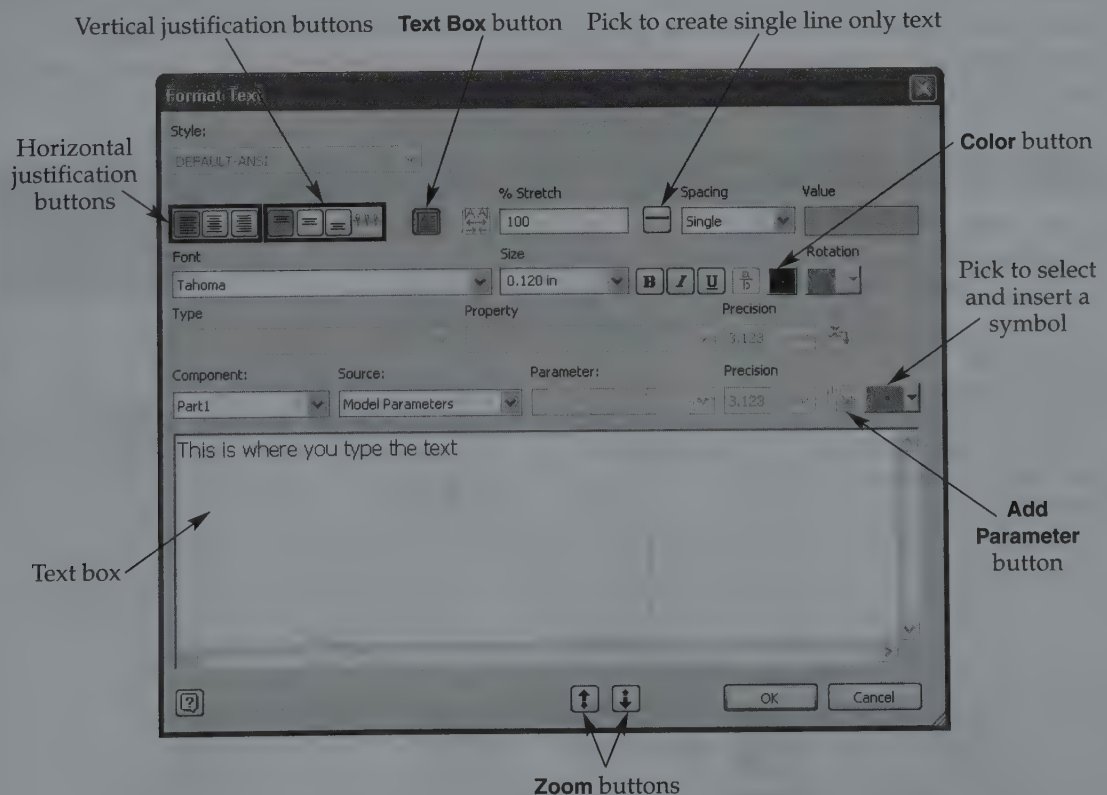
finished, pick the **OK** button to exit the dialog box. Continue placing text, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Multiline and Single-Line Text

By default, text is multiline text, with the number of lines of text based on the size of the text box and spaces between words. Pick the **Single Line Text** button to create a single line of text regardless of the size of the text box. Options specific to working with single-line text become available.

The **Line Spacing** drop-down list, available for sketching multiline text, allows you to specify the distance between multiple lines of text. Select the **Multiple** option to enable the **Value** text box and enter a smaller or greater spacing ratio, such as triple (3) or half (.5). Select the **Exactly** option to enable the **Value** text box and enter a specific distance between multiple lines of text.

Justification and Orientation

Use the **Justification** buttons to *justify* standard multiline and single-line text. The **Baseline Justification** option is available when you create single-line text. Baseline-justified text has no vertical justification. **Figure 3-32** shows standard and baseline justification options.

The **Text Box** button, selected by default, adds a construction box around text. Use the text box as you would other construction geometry, including adding constraints to define the text size and location. Once you create the text, you can toggle the text box function by right-clicking on the text and selecting or deselecting **Text Box**. If you do not use a text box, a sketch point appears according to the selected justification. Locate the point using constraints.

justify: Align the margins or edges of text. For example, left-justified text is aligned along an imaginary left border.

Display Characteristics

The **% Stretch** text box allows you to define the amount of stretch, or width, of text characters. Normal text has a value of 100. To create narrower text, enter a value less than 100. Enter a value greater than 100 for wider text. Pick the **Fit Text** button, available when sketching single-line text, to fit text into a specific space. The text width, or percent stretch, varies depending on the size of the selected space.

Use the **Font** drop-down list to define the text *font*. The **Size** drop-down list allows you to specify the text height. The format buttons define text display options. Pick the **Bold** button to create bold text, the **Italic** button to create italicized text, and the **Underlined** button to underline text.

font: A letter face design.

Figure 3-32.
Apply standard justification to multiline or single-line text, or apply baseline justification to single-line text, depending on the required text location and position.

Standard Text Justification Options

Baseline Text Justification Options

The **Stack** button allows you to stack and unstack selected text. To create a vertically stacked fraction, place a forward slash between the top and bottom items, select the text to stack, and then pick the **Stack** button, or right-click and pick **Stack**. Typing a number sign (#) between selected numbers results in a diagonal fraction bar. To apply a *tolerance stack*, type a caret (^) between the top and bottom items. **Figure 3-33** shows stack options. To unstack selected text, pick the **Stack** button again, or right-click and choose **Unstack**. To adjust stack settings using the **Stack Properties** dialog box, right-click on stacked text and pick **Properties**.

Pick the **Color** button to use the **Color** dialog box to define the text color. The **Rotation** flyout becomes enabled when you deselect the **Text Box** button, providing options for rotating the text. A 0° rotation generates text horizontally from left to right, a 270° rotation generates text vertically from top to bottom, a 180° rotation generates text horizontally from right to left, and a 90° rotation generates text vertically from bottom to top. You can specify text box rotation using constraints.

Parameters and Symbols

The function of the **Component** drop-down list in the model environment is to identify the component from which values in the **Source** and **Parameter** drop-down lists are referenced. The **Source** drop-down list specifies the type, or source, of parameters to use. You can select *model parameters* or *user parameters*. The **User Parameters** source option has no function if you do not create user parameters.

The **Parameter** drop-down list contains all the parameters available in the selected source. The parameters correspond to the parameters used to define model geometry. Use the **Precision** drop-down list to specify the precision of the displayed value. Once you select a parameter and define parameter options, pick the **Add Parameter** button to add the parameter to the text box.

To add a symbol to the text box, select a symbol from the **Symbols** flyout. If needed, pick the **Character Map** button to copy and paste a symbol from the **Character Map** dialog box.

NOTE

Pick the **Zoom In** or **Zoom Out** buttons to zoom in or out on the text in the text box. Zooming in the text box changes the *displayed* size of text in the text box, not the specified text height.

Exercise 3-12

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-12.

Figure 3-33.
Different types of character stacking. ASME standards recommend that the height of stacked fraction numerals be the same as the height of other numerals.

	Selected Text	Stacked Text
Vertical Fraction	1/2	$\frac{1}{2}$
Tolerance Stack	1^2	$\frac{1}{2}$
Diagonal Fraction	1#2	½

Using the Geometry Text Tool

The **Geometry Text** tool provides the ability to align text with an existing sketch object. Pick the object to align the text with to display the **Geometry-Text** dialog box. See **Figure 3-34**. Format text and insert content using the functions in the upper portion of the dialog box. Type text and apply associated operations such as copying and pasting in the large text box. When you are finished, pick the **OK** button to exit the dialog box. Continue placing geometry text, or press [Esc], right-click and pick **Done**, or access another tool to exit.

Many of the same options are available in the **Geometry-Text** and **Format Text** dialog boxes. The unique options in the **Geometry-Text** dialog box allow you to control the display and orientation of the text in reference to the selected geometry. **Figure 3-35**
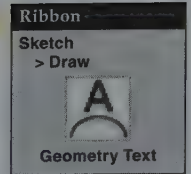
Figure 3-34.
The **Geometry-Text** dialog box appears after you pick an object to align the text with, such as the construction circle shown.
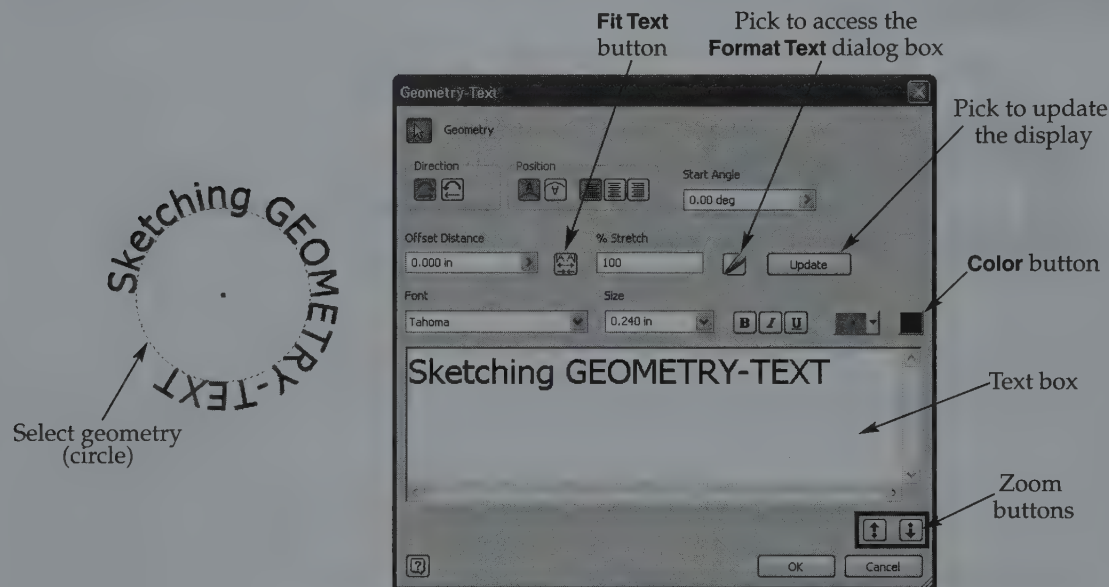
Figure 3-35.
Examples of adjustments you can make to the direction, position, and display when you sketch geometry text.

Direction: Clockwise	**Direction:** Clockwise	**Direction:** Clockwise	**Direction:** Counterclockwise
Position: Outside	**Position:** Outside	**Position:** Outside	**Position:** Inside
Justification: Left	**Justification:** Left	**Justification:** Center	**Justification:** Center
Start Angle: 0°	**Start Angle:** 90°	**Start Angle:** 90°	**Start Angle:** 0°
Offset: 0	**Offset:** Greater than 0	**Offset:** 0	**Offset:** 0
(Default settings)		**Fit Text**	

shows examples of using different geometry text characteristics to align the same text with a circle. Study this figure as you apply geometry text options. As you make changes to geometry text settings, pick the **Update** button to see the effects of the changes without exiting the dialog box.

Pick the **Clockwise** or **Counterclockwise** button in the **Direction** area to specify the direction the text travels along the selected object. Pick the **Outside** button to position the text on the outside of the selected object, or pick the **Inside** button to flip the text over the object. The selected **Justification** button determines the location of the justification point based on the start angle specified in the **Start Angle** text box.

Use the **Offset Distance** text box to define how far from the selected object the text occurs. Pick the **Launch Text Editor** button to display the **Format Text** dialog box, where you can further control text characteristics.

> **NOTE**
>
> Use the **Geometry** button to redefine the object to which text aligns.

Exercise 3-13

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 3-13.

Chapter Test

1. What is a profile?
2. Describe the purpose of a path.
3. What are the two basic steps in sketch development?
4. Why are sizes often approximate in initial sketch geometry?
5. What is the difference between a closed loop and an open loop?
6. What is a 2D plane?
7. Define *coordinate system*.
8. Where is the center point of the model coordinate system?
9. Identify the plane on which a sketch forms by default when you start a new part model and explain its rotation.
10. Name the tool used when you finish creating a sketch and are ready to enter the feature work environment.
11. Define the terms *zooming out* and *zooming in*.
12. Explain how to zoom out and in using the mouse wheel.
13. Explain what panning is and how to use the mouse to pan.
14. Briefly describe how to project the center point in the **Origin** folder onto the current sketch plane.
15. What are geometric constraints?
16. Give at least two common uses of geometric constructions.
17. What is an inferred geometric constraint?
18. Which geometric constraint specifies that objects such as lines and ellipse axes never intersect, no matter how long they become?
19. Which geometric constraint defines a 90° angle between objects such as lines and ellipse axes?
20. What is a coincident constraint? Give an example.
21. Briefly describe how to create a line with a tangent constraint.
22. What is a spline?
23. Which tool should you use to create a circle that touches three different lines at one point only?
24. What three items do you need to specify to create an ellipse using the **Ellipse** tool?
25. What is a regular polygon?
26. Explain the difference between rounds and fillets.
27. Briefly describe how to add an equal-distance chamfer to a sketch.
28. What is the difference between sketch points and center points?
29. Name the tool used to add a basic block of text to a sketch.
30. Briefly describe how to align sketch text with a curved surface, such as a circle.

Problems

Instructions:

- *Create sketches of the following objects using the techniques described in this chapter, zooming and panning as necessary.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate, but do not place additional geometric constraints.*
- *Use the information in the status bar to create objects at the approximate size given by the dimensions, but do not add dimensional constraints.*
- *Add as much information as possible to the **iProperties** dialog box. Do not assign material and color properties at this time.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **Title:** PIN
   Units: Metric
   Template: Part-mm.ipt
   Part number: IAA-001-04
   Description: C-CLAMP PIN
   Project: C-CLAMP
   Sketch plane: YZ
   Save as: P3-1.ipt
   Specific Instructions: Position the center of the circle at the projected center point by inferring a coincident constraint.

4.75

Projected
center point
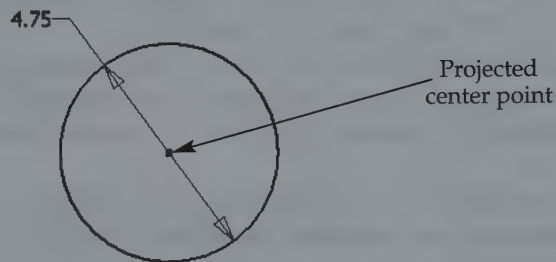
2. **Title:** SWIVEL
   Units: Metric
   Template: Part-mm.ipt
   Part number: IAA-001-02
   Description: C-CLAMP SWIVEL
   Project: C-CLAMP
   Sketch plane: XZ
   Save as: P3-2.ipt
   Specific instructions: Position the center of the circle at the projected center point by inferring a coincident constraint.
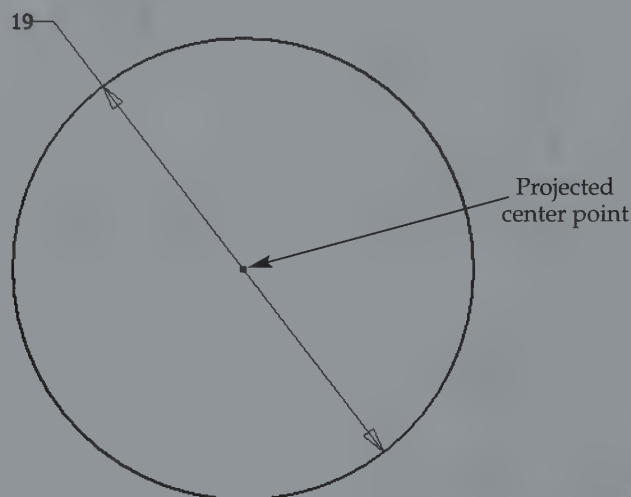
3. **Title:** SCREW
   Units: Metric
   Template: Part-mm.ipt
   Part number: IAA-001-03
   Description: C-CLAMP SCREW
   Project: C-CLAMP
   Sketch plane: XZ
   Save as: P3-3.ipt
   Specific instructions: Position the center of the circle at the projected center point by inferring a coincident constraint.
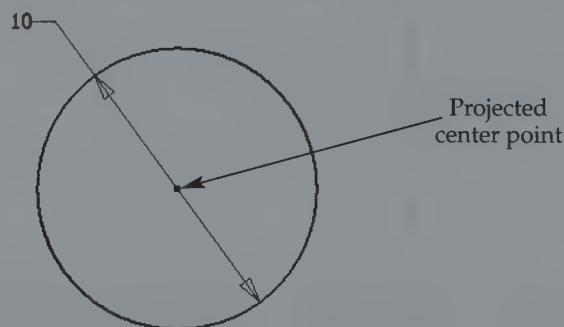
4. **Title:** SUPPORT BRACKET
   Units: Inch
   Template: Part-IN.ipt
   Part number: IAA-002-01
   Sketch plane: XY
   Save as: P3-4.ipt
   Specific instructions: Position the center of the .500 circle at the projected center point by inferring a coincident constraint.
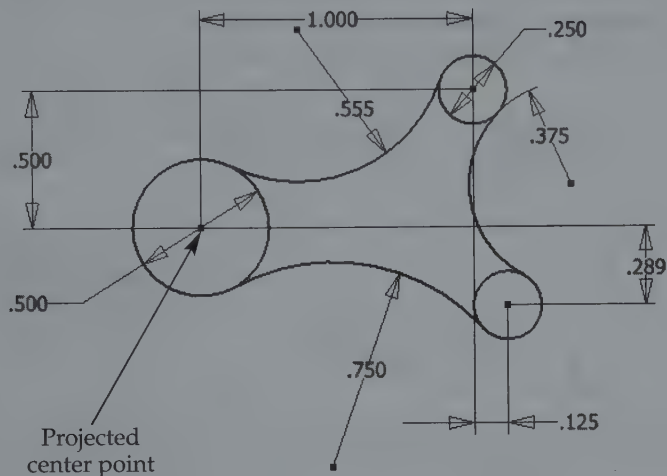
5. **Title:** SUPPORT BLOCK
   Units: Inch
   Template: Part-IN.ipt
   Part number: IAA-003-01
   Sketch plane: XZ
   Save as: P3-5.ipt
   Specific instructions: Position the lower-left corner of the sketch at the projected center point by inferring a coincident constraint.

6. **Title:** BODY
   Units: Metric
   Template: Part-mm.ipt
   Part number: IAA-001-01
   Description: C-CLAMP BODY
   Project: C-CLAMP
   Sketch plane: XY
   Save as: P3-6.ipt
   Specific instructions: Position the sketch at the projected center point by inferring a coincident constraint as shown.
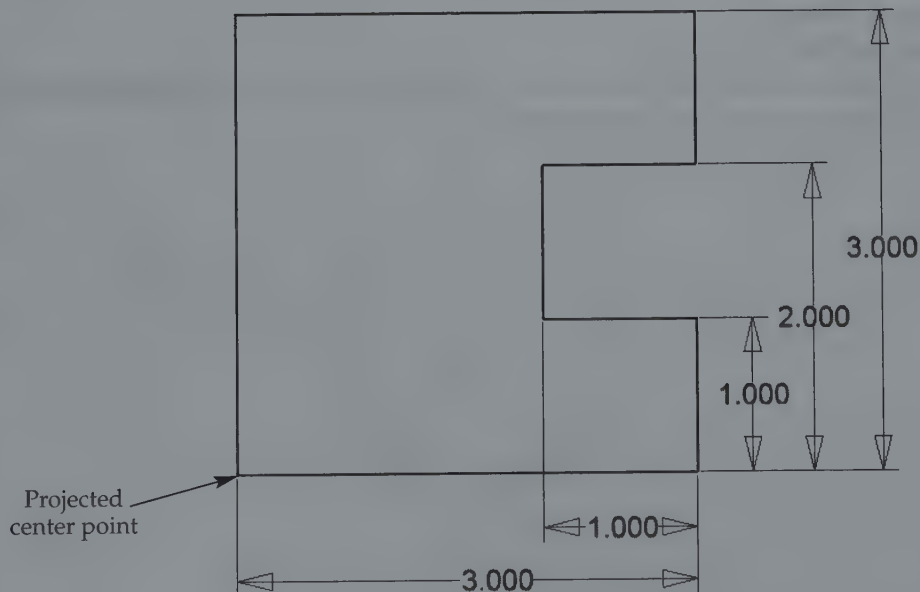
7. **Title:** HEAD
   Units: Inch
   Template: Part-IN.ipt
   Part number: IAA-004-01
   Project: PUSH PIN
   Sketch plane: XY
   Save as: P3-7.ipt
   Specific instructions: Position the sketch at the projected center point by inferring a coincident constraint as shown. Do not begin the sketch directly on the projected center point.
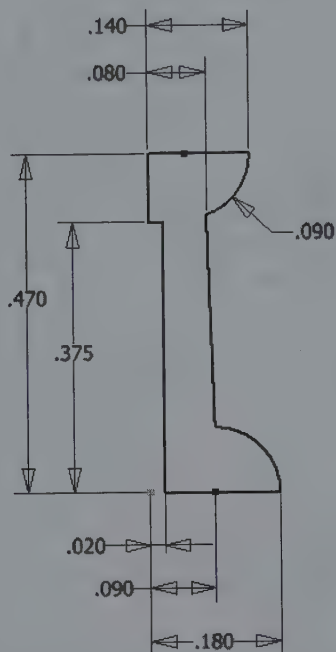
8. Draw a freehand sketch of the 3D indicator. Label the X, Y, and Z axes and specify the color of each. Describe the purpose of the 3D indicator.

Refining Sketches

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Add geometric constraints to existing sketch objects.
✓ Add dimensional constraints.
✓ Pattern and offset sketch geometry.
✓ Explain the function of the **Inventor Precise Input** toolbar.
✓ Use sketch editing tools and techniques.
✓ Work with sketch parameters.
✓ Troubleshoot sketches using the Sketch Doctor.

This chapter describes tools and options for finalizing a sketch. You will add dimensional constraints to size and locate sketch objects. You will also use other sketch geometry tools and explore methods of editing a sketch. Once you prepare a sketch or group of sketches, you are ready to build a sketched feature.

Geometric Constraints

Use geometric constraints to establish geometric constructions and relationships. For example, apply an equal constraint to two circles if it is necessary for the diameter of the circles to remain the same throughout the design process. Then dimensionally constrain one of the circles. If you modify the dimension, the equal-constrained circle automatically adjusts also.

Adding Geometric Constraints

Inferring geometric constraints is usually the most effective way to begin constraining a sketch. For basic sketch shapes, inferred geometric constraints may be the only geometric constraints necessary. Add geometric constraints to existing objects when geometric constraints do not infer, or after deleting geometric constraints or constrained objects. A combination of inferred and additional geometric constraints is often required.

Access geometric constraints from the **Constraints** panel of the **Sketch** ribbon tab, or right-click and select from the **Create Constraint** cascading menu. **Figure 4-1**

Figure 4-1.
Use geometric constraint tools to form geometric constructions and help constrain a sketch.

Constraint Tool	Icon	Description
Perpendicular		Forms a perpendicular construction, or 90° angle, between lines or ellipse axes. Unless the constraint is removed, you cannot increase or decrease the angle between the objects.
Parallel		Creates a parallel construction. Objects such as lines and ellipse axes never intersect, no matter how long they become.
Tangent		Defines the tangent edge of one curve, such as a line, spline, circle, arc, or fillet, in reference to the tangent edge of an existing curve.
Smooth (G2)		Creates a curvature-continuous situation, or G2 curve, between a spline and a line, second spline, or arc connected to the spline endpoint.
Coincident		Constrains two points to the same location, or constraints a point along a curve. A point can be coincident to a curve without touching the curve.
Concentric		Constrains the center point of an ellipse, circle, or arc to the center point of another ellipse, circle, or arc. Select objects, not points, to constrain.
Collinear		Aligns two lines or ellipse axes along the same line.
Equal		Sizes and locates an object in reference to another object. This tool is also available by typing the [=] key.
Horizontal		Horizontally aligns lines, points, or ellipse axes. Positions geometry along the X axis on the XY plane, along the Y axis on the YZ plane, and along the X axis on the XZ plane.
Vertical		Vertically aligns lines, points, or ellipse axes. Positions geometry along the Y axis on the XY plane, along the Z axis on the YZ plane, and along the Z axis on the XZ plane. This tool is also available by typing the [l] key.
Fix		Secures a point or object to its current location in space.
Symmetric		Allows you to establish symmetry by selecting one object, followed by another object, and finally, a line of symmetry.

describes each geometric constraint tool. Many geometric constraint tools apply the same constraints as those you can infer. For constraint tools that require you to pick two objects, the first object usually remains the same and the second object changes in relation to the initial object. For example, to make two lines perpendicular using the **Perpendicular** tool, select the line that remains in the same position, followed by the line to make perpendicular to the first line. This is a general rule that may not always apply because of other constraints.

To constrain more geometry using the same tool, continue selecting objects. To use a different constraint tool, access that tool and begin selecting objects. To exit the tool without activating another tool, press [Esc] or right-click and pick **Done**.

Exercise 4-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-1.

Managing Geometric Constraints

Access the **Show Constraints** tool and select objects to display constraint *glyphs* associated with the selected objects. See **Figure 4-2**. Use the **Show All Constraints** tool to show all constraint glyphs assigned to all objects in the sketch. Access the **Constraint Visibility** dialog box to specify which constraint glyphs are displayed. By default, all constraint types are shown. Select the check boxes corresponding to the constraint glyphs to show, and use the **Select All** and **Clear All** buttons to aid selection. Limiting constraint visibility to specific constraint types often helps to locate and adjust constraints.

Coincident constraints appear as dots that, when hovered over, show coincident constraint glyphs. All other constraints appear as glyphs. When you hover over or select a glyph, the corresponding constraint glyph and constrained objects are highlighted. This allows you to recognize the objects and constraints associated with the glyph. If constraint glyphs block your view, drag them to a new location.

To hide a specific glyph, pick the **Hide** button to the right of the symbols. Use the **Hide All Constraints** tool to hide all constraints shown in the sketch. To delete existing constraints in order to apply different constructions, right-click on a glyph and select **Delete**. If you delete an object, all constraints assigned to the object and related objects are also deleted.

Ribbon

Sketch
  > Constrain
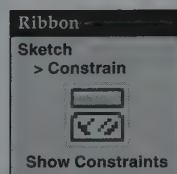
Show Constraints

glyph: A graphic representation that initiates an action, symbol, or function.

Shortcut
  Show All Constraints
Type
  [F8]

Shortcut
  Constraint Visibility...

Shortcut
  Hide All Constraints
Type
  [F9]
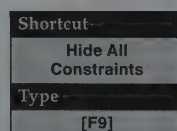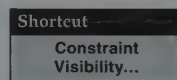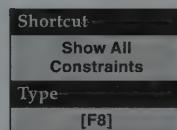
Figure 4-2.
Display geometric constraint glyphs by picking the **Show Constraints** button and then selecting the object.
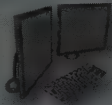
Exercise 4-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-2.

Dimensional Constraints

dimensional constraint: A measurement that numerically defines the size and location of sketch geometry, such as the length of a line, diameter of a circle, or radius of an arc.

Dimensional constraints establish size and location parameters. You must include dimensional constraints to create a truly parametric sketch. Once dimensional constraints are placed, you can view, adjust, and remove them as needed.

Using the General Dimension Tool

Ribbon

**Sketch
> Constrain**

Dimension

Shortcut

Create Dimension

Type

[D]

Access the **General Dimension** tool to add aligned, linear, diameter, radius, or angular dimensional constraints. An icon representing a possible dimensional constraint appears as you move the cursor over specific objects. Pick geometry that is appropriate for the type of dimensional constraint, as explained later in this chapter. Then select a location for the value and specify the value. Repeat the steps to assign other dimensional constraints, or press [Esc], right-click and pick **Done**, or access another tool to exit.

By default, a dimensional constraint assumes the size or location value of the existing object. Pick the dimensional constraint to open the **Edit Dimension** dialog box, where you can specify a new value to change the size or location of the object. See **Figure 4-3**. For many applications, you will find that it is more efficient to display the **Edit Dimension** dialog box immediately when you create a dimensional constraint. To do this, pick the **Edit dimensions when created** check box in the **Sketch** tab of the **Application Options** dialog box. You can also activate the **Edit dimensions when created** option while using the **General Dimension** tool by right-clicking *before* you locate a dimensional constraint, and selecting **Edit Dimension**.
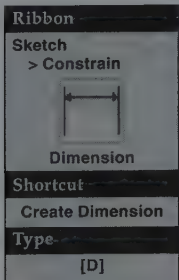
Figure 4-3.
The **Edit Dimension** dialog box allows you to specify a dimensional constraint value. The dialog box appears automatically when you place the dimension if you select the **Edit Dimension** option of the **General Dimension** shortcut menu.

Pick when finished

When you add a dimension, the **Edit Dimension** dialog box will appear if the **Edit Dimension** check box is on

---
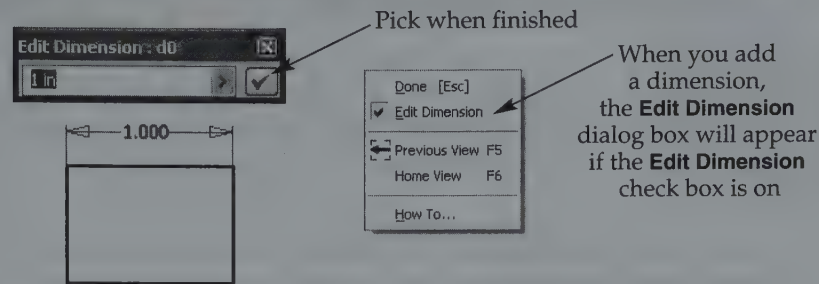
NOTE

To edit dimensional constraints at any time, even while the **General Dimension** tool is not active; double-click a dimensional constraint to access the **Edit Dimension** dialog box.

There are several ways to specify dimensional constraint values in the **Edit Dimension** dialog box. The most basic options are to type a value or enter an equation in the text box. Dimensional constraint units reflect the current work environment and unit settings. For example, typing 1 means 1 inch when you are working with inch units. However, you can enter units after a numerical value to convert the value to the specified units. For example, you can enter 20 mm while working in an inch-unit system.

Equations are common for linking a dimensional constraint value to an existing dimensional constraint, which enables the sketch to adapt according to parameter changes. Equations are also used when the exact numerical dimensional constraint value is unknown. **Figure 4-4** shows some keys used to create basic expressions. In addition to these standard operations, you can use functions such as sin, cos, log, and tan.

NOTE

The letters fx: before the value differentiates a dimensional constraint value specified using an equation from other dimensional constraints. For example, an equation of 2+2 appears as fx:4 in the sketch.

Figure 4-4.
Keyboard keys used to create basic expressions to specify dimensional constraint values. Many other expressions are also possible.

Key	Function	Example
+	add	2+2 or 3in+5mm
–	subtract	5–2 or 7in–5mm
*	multiply	5*2 or 7in*5mm
/	divide	6/2 or 7in/5mm
^	power operation	3^5
()	separate expressions	(5*3)+(47.25/26)
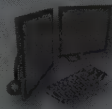
Another way to specify a dimensional constraint value is to reference existing sketch or model information while using the **Edit Dimension** dialog box. One method is to pick **Measure** from the text box menu and then pick an object, such as a line, or two points to add the length of the object or distance between the two points to the text box. You can also select recently entered values from the text box menu. To access the text box menu, pick the arrow to the right of the value shown in the text box.

Another, often more useful, technique is to reference a sketch or model parameter. If you know the name of a parameter, enter the name in the text box. A quick way to use a sketch parameter without knowing the parameter name is to pick a dimensional constraint to copy the parameter to the text box. See **Figure 4-5**. If you pick the check mark, the object becomes the same size as the reference object. This is essentially the same constraint that occurs with an equal geometric constraint. As a result, when you reference an existing dimensional constraint, you usually include an equation, as shown in **Figure 4-6**.

Figure 4-5.
When adding a new dimensional constraint, pick an existing dimensional constraint to create a reference.

Referenced dimension

After you close the **Edit Dimension** dialog box, the dimension and geometry change to match the referenced dimension
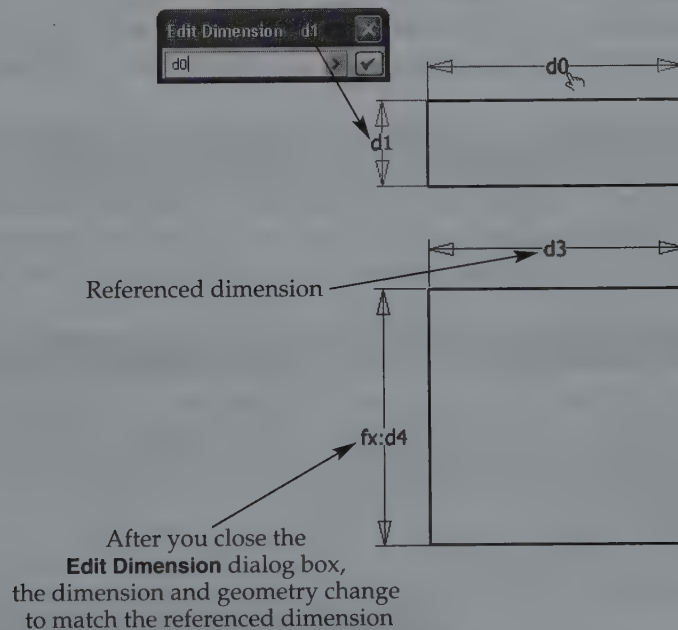
Figure 4-6.
Using existing dimensional constraints and equations to define an object.

Equation (*ul* means "unitless" and is applied to values that do not contain specified units)

Result of equation (d1)
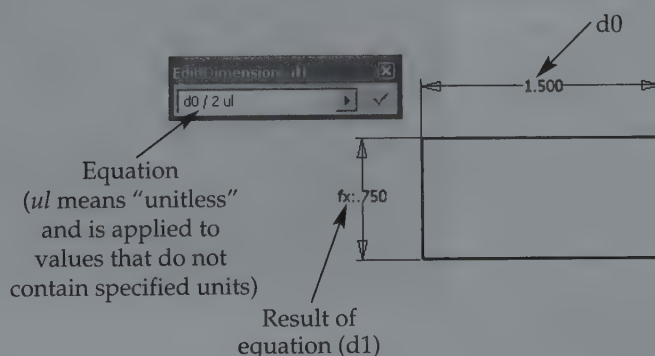
Linear

To place a *linear dimensional constraint*, pick one or two curves or points. Many dimensional constraints, especially linear dimensional constraints, originate from surfaces. Select the actual surfaces from which the measurement originates when possible. For example, in **Figure 4-7**, pick the two vertical lines to specify the 1.200 value, not the endpoints or the diagonal line.

If the object is vertical, drag the dimension to the left or right and pick a location. If the object is horizontal, drag the dimension up or down and pick a location. When it is difficult to form the appropriate alignment, right-click and select **Vertical** or **Horizontal** before placing the value to help isolate a vertical or horizontal format.

When applying linear dimensional constraints to circular objects, you have the option of selecting the center of the circular object or a tangent quadrant. Circular features are usually located according to the center point, which you can select by picking a center point or a curve. See **Figure 4-8A**. Look for the icon shown in **Figure 4-8B** to reference a

linear dimensional constraint: A type of dimensional constraint used to define the vertical and horizontal size and location of object features.

Figure 4-7.
Linear sketch dimensional constraints control horizontal and vertical sizes and locations.

Right-clicking while placing .350 dimension produces this shortcut menu

| Done [Esc] |
| Linear Diameter |
| Edit Dimension |
| ⬅ Previous View F5 |
| Home View F6 |
| How To... |

1.200   .350   .750

| Done [Esc] |
| Aligned |
| Vertical |
| Horizontal |
| Edit Dimension |
| ⬅ Previous View F5 |
| Home View F6 |
| How To... |

Right-clicking while placing the .750 or 1.200 dimension produces this shortcut menu because this dimension can only be vertical

---

Figure 4-8.
A—An example of linear dimension constraining to the center of a circular object. B—You can also constrain to the tangent edge of a circular object when necessary.

Pick second to dimension to the center point
Linear icon
.500
Pick first
A

Pick second to dimension to the tangent edge
Tangent icon
.750
Pick first
B

---
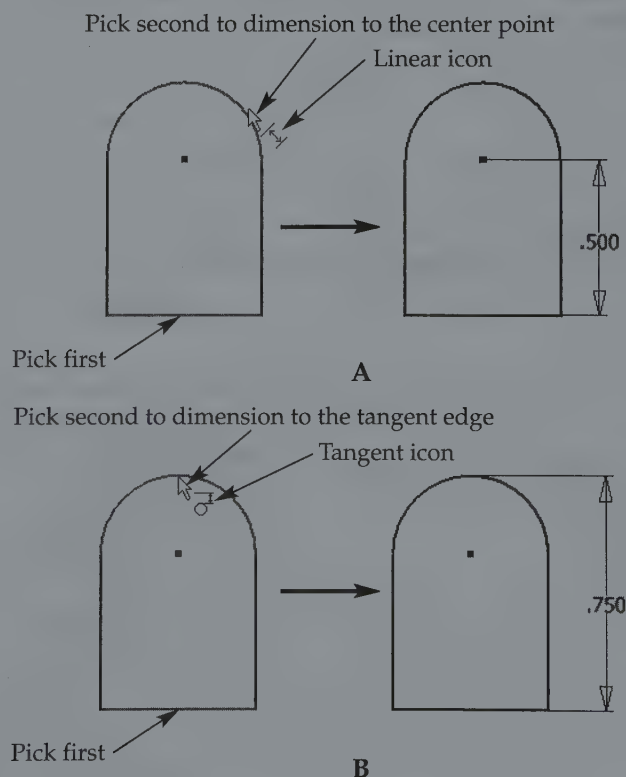
tangent quadrant before picking. This option is common for construction purposes and occasionally for some drafting applications.

linear diameter: A diameter shown in an edge view, so that the diameter appears as a straight line.

Aligned

To place an *aligned dimensional constraint,* pick one or two curves or points. If you select a single angled line, the default format is linear. To use an aligned format, select the object a second time or right-click and pick **Aligned**. Then pick a location for the value. See **Figure 4-9**. When creating an aligned dimensional constraint, you can pick a center point or a tangent edge, similar to the linear applications shown in **Figure 4-8**.

aligned dimensional constraint: A dimensional constraint used to define an object or feature that is not vertical or horizontal.

Angular

To place an *angular dimensional constraint,* select two lines followed by a location for the value. See **Figure 4-10**. The location, or quadrant, you select when placing the value determines the angle measured. Four placements are possible.

angular dimensional constraint: A dimensional constraint used to define the angle between two lines.

Diameter and Radius

To add a *diameter dimensional constraint* to specify the *diameter* of a circle, pick a circle and a location for the value. To constrain a circle using the *radius,* select the circle, right-click, and choose **Radius**. See **Figure 4-11**. To add a *radius dimensional constraint,* to specify the radius of an arc, pick an arc and a location for the value. To constrain an arc using the diameter, select the arc, right-click, and choose **Diameter**. See **Figure 4-12**.

diameter dimensional constraint: A dimensional constraint used to define the diameter of a circle or circular object.

diameter: The distance across a circle from one side to the other through the center.

radius: The distance from the center of a circle or arc to its circumference.

radius dimensional constraint: A dimensional constraint used to define the radius of an arc or circular feature.

Figure 4-9.
Two methods to apply an aligned dimensional constraint.

Pick the object twice to convert to an aligned format, or ...

Right-click and pick **Aligned** to add an aligned dimension

1.250

Done [Esc]
Aligned
Vertical
Horizontal
Edit Dimension

Previous View F5
Home View   F6

How To...

Aligned icon
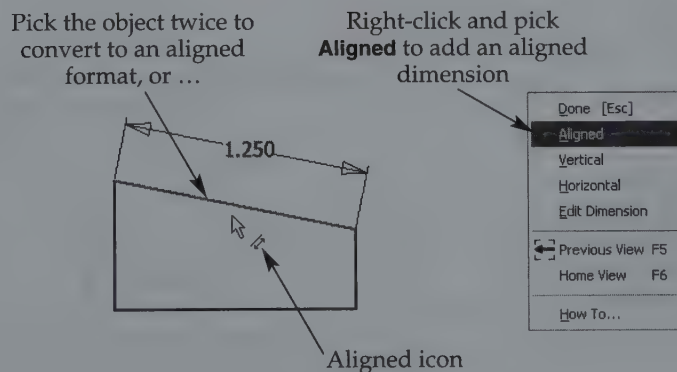
Figure 4-10.
Applying an angular dimensional constraint requires two nonparallel lines.

Pick the two lines and place the dimension

Angular icon

65.00
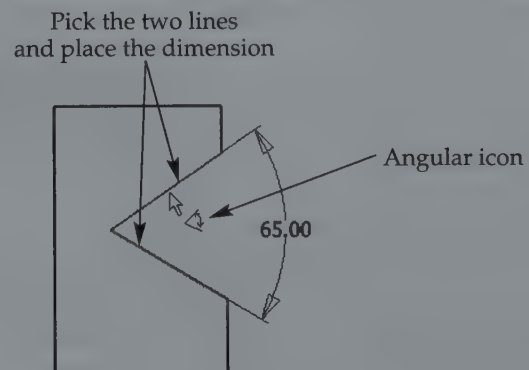
Figure 4-11.
Dimensionally
constrain circles
using the diameter,
except in some
construction
applications when
a radius is more
appropriate.

Pick the circle and pick to
place the diameter dimension

Diameter
icon

.750

Right-click before locating the
dimension value and pick **Radius**
to place a radius dimension

.375

Done [Esc]
Radius
Edit Dimension
Previous View F5
Home View   F6
How To...
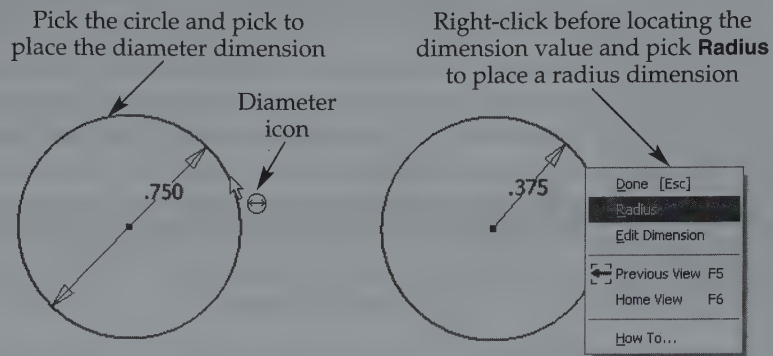
Figure 4-12.
Dimensionally
constrain arcs using
the radius, except in
some construction
applications when
a diameter is more
appropriate.

Pick the arc and pick to
place the radius dimension

.500   Radius icon

1.000

Done [Esc]
Diameter
Edit Dimension
Previous View F5
Home View   F6
How To...

Right-click before locating the
dimension value and select **Diameter**
to place a diameter dimension
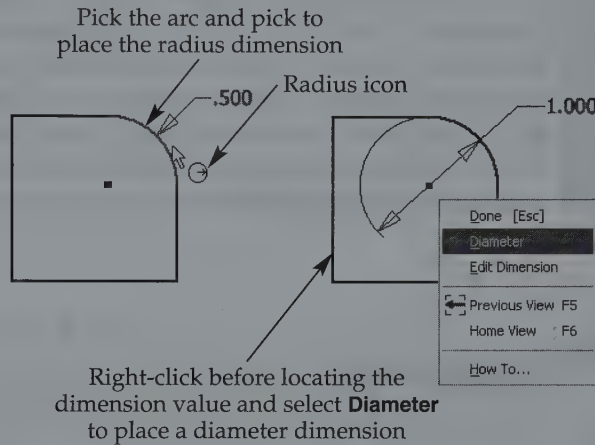
Driven Dimensions

When you add too many dimensional constraints, the sketch should become over-constrained. However, Inventor does not allow over-constraining to occur. This reinforces a fundamental dimensioning rule, does not allow the buildup of tolerances, and preserves design intent. As a result, you can either not accept the dimensional constraint and cancel the tool, or allow the dimensional constraint to become a *driven dimensional constraint*. See **Figure 4-13**.

You cannot edit a driven dimensional constraint to change the size of an object, but a driven dimensional constraint changes if the corresponding parameters change. To prevent Inventor from warning you of an over-constrained condition, pick the **Apply Driven Dimension** radio button in the **Overconstrained Dimensions** area of the **Sketch** tab in the **Application Options** dialog box.

driven dimensional constraint: A dimensional constraint used for reference purposes only. Parentheses enclose reference dimensional constraints to show that they are driven.

Figure 4-13.
The driven
dimensional
constraint, identified
by the parentheses,
is for reference only
and is not directly
editable.

1.000

.200

.350

Driven
dimension

(.550)
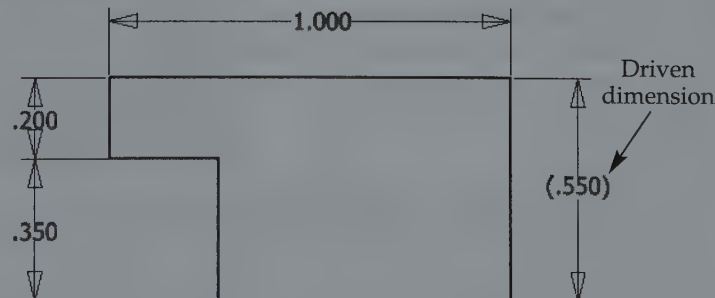
PROFESSIONAL TIP

Driven dimensional constraints are sometimes appropriate for reference, especially in creating an adaptive part. A non-driven dimensional constraint uses the **Normal** style. To change a non-driven dimensional constraint to a driven dimensional constraint, pick the **Driven Dimension** button before constraining, or select existing dimensional constraints and then pick a button to convert the format.

NOTE

Dimensional constraint appearance does not comply with ASME standards. Do not be overly concerned about the placement or display characteristics of dimensional constraints, but if possible, apply dimensional constraints just as you would add dimensions to a drawing using correct drafting practices. In addition, move and manipulate dimensional constraints so the sketch environment is as uncluttered as possible. Select and drag a dimensional constraint to move it.

Exercise 4-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-3.

Using the Auto Dimension Tool

The **Auto Dimension** tool allows you to constrain several objects at once using the **Auto Dimension** dialog box shown in **Figure 4-14**. The **Dimensions Required** display box identifies the number of constraints that may be required to constrain the sketch. If you immediately pick the **Apply** button, all possible geometric and dimensional constraints are applied to the sketch. An alternative is to use the **Curves** button, which is active by default, to select specific geometry to constrain.

Figure 4-14.
You can use the **Auto Dimension** dialog box to add geometric and dimensional constraints and to remove all dimensional constraints. The **Dimensions Required** display box provides a good way to determine the possible number of constraints required to fully constrain the sketch.

Choose to apply dimensions, constraints, or both automatically to fully constrain the sketch

Auto Dimension

Curves   ☑ Dimensions
          ☑ Constraints

4 Dimensions Required

[?] Apply   Remove   Done
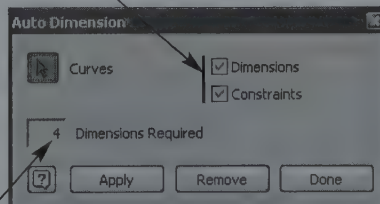
Number of dimensions needed to fully constrain the sketch

Deselect the **Dimensions** check box to avoid adding dimensional constraints, or deselect the **Constraints** check box to avoid forming geometric constraints. The **Auto Dimension** dialog box includes a **Remove** button for efficiently removing all dimensional constraints that have been added to the sketch. Pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

NOTE

The dimensional constraints you create using the **General Dimension** tool are not replaced when you use the **Auto Dimension** tool.

PROFESSIONAL TIP

The **Edit dimensions when created** check box on the **Sketch** tab of the **Application Options** dialog box has no function in the **Auto Dimension** tool. Therefore, dimensional constraints placed using the **Auto Dimension** tool represent the current object measurements. To modify a dimensional constraint value, double-click on the dimensional constraint and enter a value in the **Edit Dimension** dialog box.

Exercise 4-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-4.

Supplemental Material *Dimensional Constraint Properties*

For information about controlling dimensional constraint properties, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Dimensional Constraint Properties**.

Patterning Sketch Geometry

A *sketch pattern* is useful for some applications, such as when you are sketching symmetrical geometry or when the sketch should control patterned features. However, often feature patterns are more appropriate than sketch patterns. As you will learn, creating a sketch pattern is almost identical to creating a feature pattern.

sketch pattern: Multiple arranged copies, or a pattern, of sketch shapes.

Mirroring

Access the **Mirror** tool to mirror sketch geometry across a line of symmetry using the **Mirror** dialog box. See **Figure 4-15**. Sketch a line of symmetry, typically using the centerline or centerline and construction linetype, before using the **Mirror** tool. The **Select** button is active by default, allowing you to select objects to mirror. Use window, crossing, or [Ctrl] selection to select multiple objects. Next, pick the **Mirror line** button and select the line of symmetry. Pick the **Apply** button to create the mirror. Continue

Figure 4-15.
Use the **Mirror** tool and a line of symmetry to make a mirrored copy of objects.
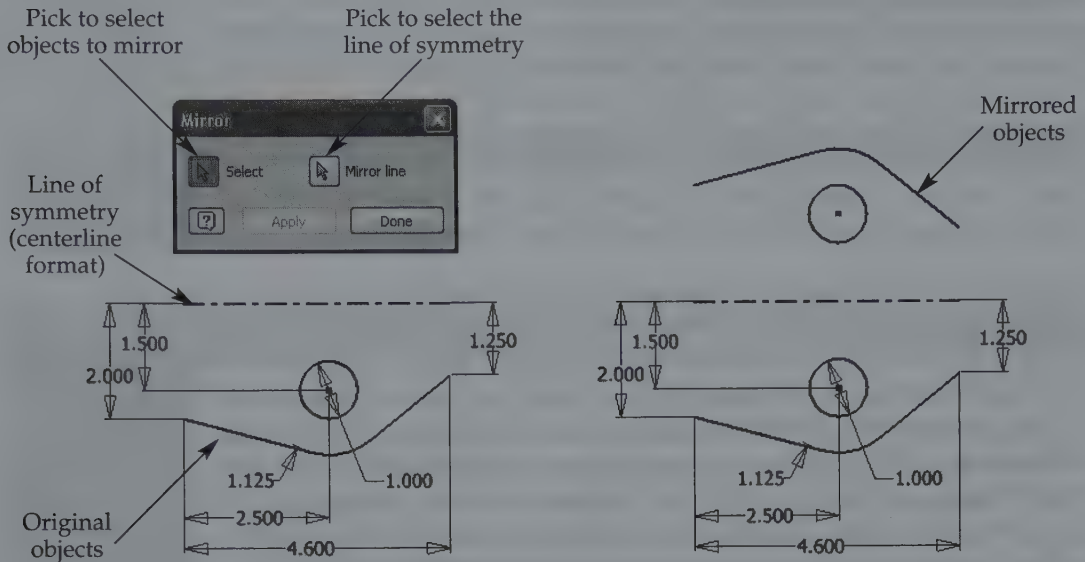
mirroring objects, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

> **NOTE**
>
> The **Mirror** tool maintains the mirrored relationship between objects by assigning symmetric constraints to the original objects and mirrored pattern.

Exercise 4-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-5.

Rectangular Patterns
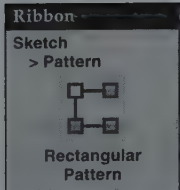
occurrence: The total number of items (sketch objects or features) patterned.

Access the **Rectangular Pattern** tool to create a rectangular pattern of sketch objects using the **Rectangular Pattern** dialog box. See **Figure 4-16.** The **Geometry** button is active by default, allowing you to select objects to pattern. Pick the **Direction 1** button, and select a curve to specify the first pattern direction. You may have to sketch a construction line to select a direction, as shown in **Figure 4-16A.** If the preview and direction arrow appear incorrect, pick the **Flip** button to reverse the direction.

Next, specify the total number of items, or *occurrences,* using the **Count** text box. The **Fitted** check box is deselected by default, allowing you to define the distance between copies in the **Spacing** text box. Spacing between copies is the distance from a point on one object to the corresponding point on the next object, not the clear space between items. For example, to pattern a hexagon that is 1″ (25 mm) across the flats and leave a .5″ (12.5 mm) space between pattern copies, specify a 1.5″ (37.5 mm) spacing. See **Figure 4-16B.** If you select the **Fitted** check box, the count divides equally within the value you specify in the **Spacing** text box. See **Figure 4-16B.**

Figure 4-16.
A—Using the **Rectangular Pattern** dialog box to create a rectangular pattern of polygons.
B—Patterning the polygon using a space between occurrences, and fitting the count within a specified distance.
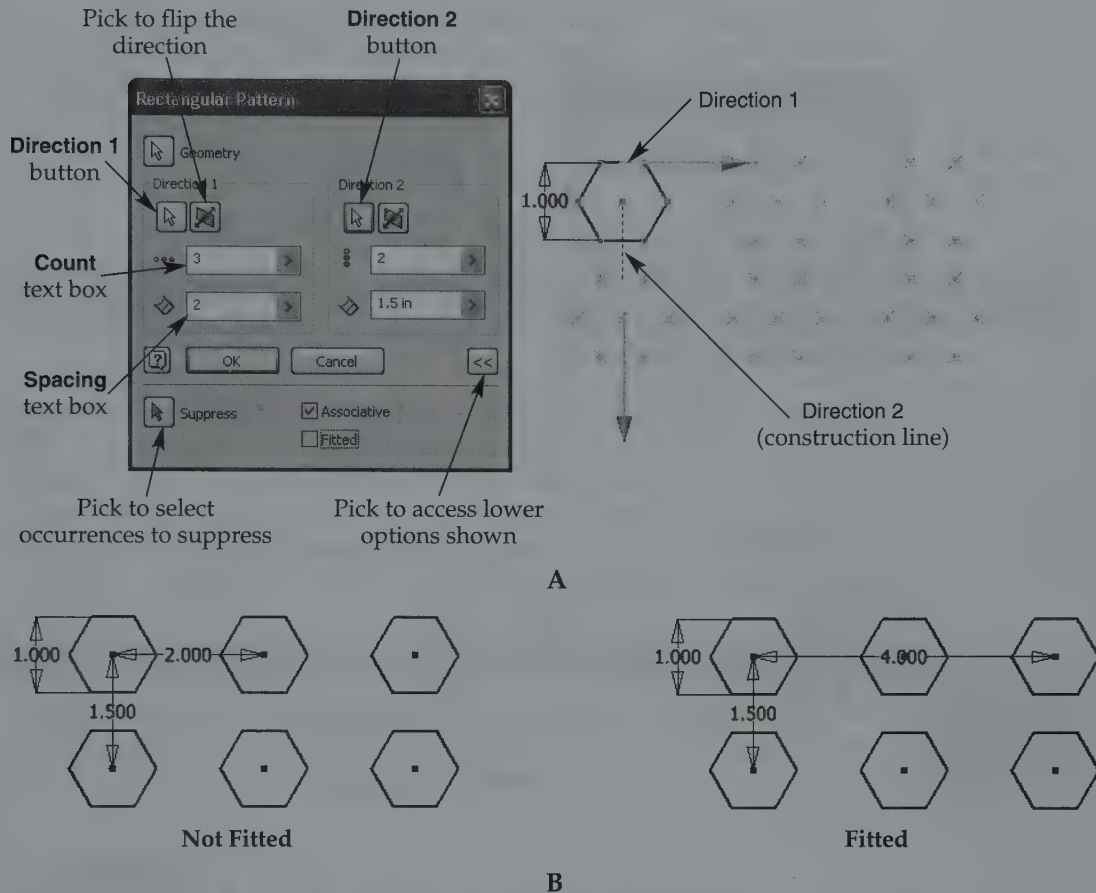
A

Not Fitted   Fitted

B

After you define Direction 1, repeat the steps to define Direction 2. For most applications, the second direction is not parallel to the first direction. Additional pattern options are available in the lower portion of the dialog box. Pick the **Suppress** button and then select pattern occurrences to exclude from the pattern. Select the **Associative** check box to make the pattern associative, which means that when you make changes to patterned geometry, the entire pattern automatically updates to reflect the changes. Deselect the **Associative** check box to have the ability to modify each occurrence individually. Pick the **OK** button to create the pattern.

Circular Patterns

Access the **Circular Pattern** tool to create a circular pattern of sketch objects using the **Circular Pattern** dialog box. See **Figure 4-17**. The **Geometry** button is active by default, allowing you to select objects to pattern. Pick the **Axis** button and select a point to specify the *axis of rotation*. You may have to sketch a point or center point to select an axis, as shown in **Figure 4-17A**. If the preview and rotation arrow appear incorrect, pick the **Flip** button to reverse the rotation.

Ribbon

Sketch
> Pattern
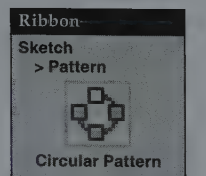
Circular Pattern

axis of rotation:
The axis around which the selected geometry is rotated or copied.

Figure 4-17.
A—Using the **Circular Pattern** dialog box to create a circular pattern of polygons. B—Patterning the polygon by fitting the count within a specified distance, and using a space between occurrences.

A

Fitted   Not Fitted

B
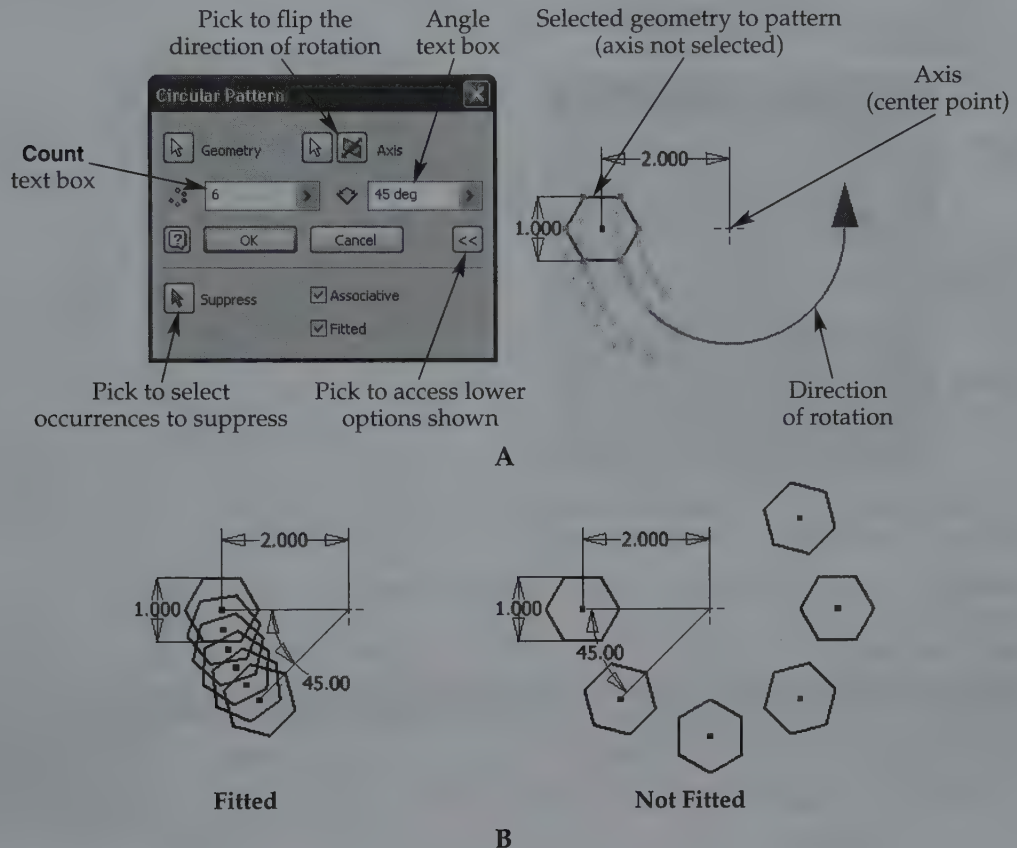
Next, specify the number of occurrences using the **Count** text box. The **Fitted** check box is selected by default, allowing you to divide the count equally within the included angle specified in the **Angle** text box. See **Figure 4-17B**. Deselect the **Fitted** check box to specify the included angle between each occurrence. See **Figure 4-17B**. Pick the **Suppress** button and then select pattern occurrences to exclude from the pattern. Select the **Associative** check box to make the pattern associative. Pick the **OK** button to create the pattern.

> **NOTE**
>
> Dimensional constraints correspond to pattern specifications, and pattern geometric constraints link occurrences. Edit the dimensional or geometric constraints to make changes to the pattern. Right-click on a patterned object to access options for suppressing elements and for editing or deleting the pattern. You must delete a pattern to remove associated objects. Suppressed pattern occurrences appear as construction geometry.

Exercise 4-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-6.

Figure 4-18.
The **Offset** tool allows you to create parallel objects.

Original geometry   Offset geometry
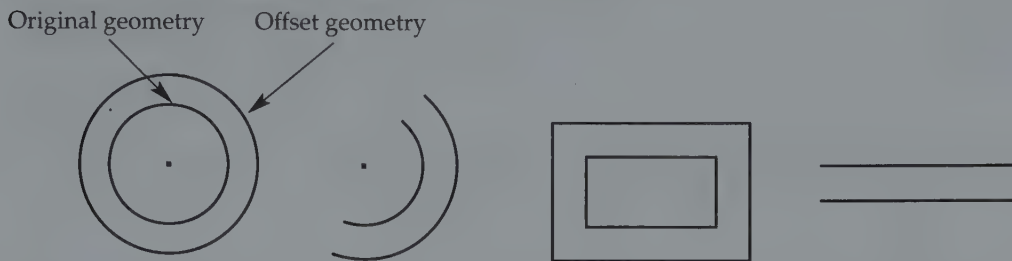
Offsetting Sketch Geometry

Access the **Offset** tool to *offset* sketch geometry. Select the objects to offset and pick the location, or offset distance. Notice that the offset distance appears on the right side of the status bar for reference. As shown by the rectangle in **Figure 4-18**, an entire loop is selected by default when you offset. To select individual elements, such as a single line in the rectangle, right-click and deselect **Loop Select**. Pick the objects to offset and then right-click and pick **Continue** or press [Enter]. Finally, pick the offset location. Continue offsetting, or press [Esc], right-click and pick **Done**, or access another tool to exit.

offset: Form objects parallel to the specified geometry at a specified distance apart.

Ribbon
Sketch
> Modify

Offset

Type

[O]

NOTE

By default, offsets receive the same constraints as the parent objects. In order to avoid applying the parent object constraints to the offset, right-click and deselect **Constrain Offset**.

Exercise 4-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-7.

Precise Input

The **Inventor Precise Input** toolbar, shown in **Figure 4-19**, allows you to identify the initial size and location of sketch geometry while sketching. For example, you can

Ribbon
Sketch
> Draw
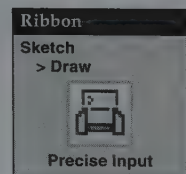
Precise Input

Figure 4-19.
The **Inventor Precise Input** toolbar allows you to locate X, Y, and Z points while sketching.

Precise Redefine button

Precise Delta button   Input Type drop-down list

Reset to Origin button   Input boxes
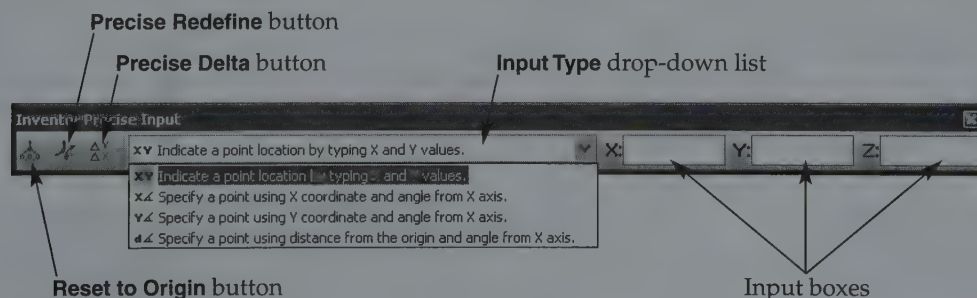
define the exact start or endpoint of a line or the center point of a circle. To access the **Inventor Precise Input** toolbar, you must first activate a sketch tool, such as **Line**.

Precise input does not constrain sketch geometry. In order to constrain the sketch, you must place geometric and dimensional constraints. As a result, in most cases, sketching with precise input takes longer and is more complicated than sketching objects and then placing dimensional and geometric constraints. Still, the **Inventor Precise Input** toolbar can be effective for some applications, and it is available for specialty viewing and sketch drawing and modification tools.

Editing Sketches

A variety of tools and options are available for adjusting existing sketch geometry. For example, you can delete or add objects, or edit or remove geometric and dimensional constraints. Specific sketch modification tools are also available.

> **NOTE**
>
> The appearance of sketch geometry is set according to the color scheme specified in the **Color Scheme** area in the **Color** tab of the **Application Options** dialog box. To adjust specific sketch geometry color, linetype and scale, and lineweight, use options available from the expanded **Format** panel of the **Sketch** ribbon tab or the **Sketch Properties** toolbar, or right-click on objects and choose **Properties...** to use the **Geometry Properties** dialog box.

Ribbon
Sketch
> Format

Sketch Properties

Analyzing Constraints

To confirm that a geometric or dimensional constraint is present and appropriate, attempt to drag a point or curve. See **Figure 4-20**. As a sketch becomes constrained, you should observe less freedom of movement. Dragging or attempting to drag objects is one of the fastest and most effective ways to analyze where a constraint is still required and to assess design options. You know when a sketch is constraining because the object color changes and you are no longer able to drag geometry.

Figure 4-20.
The drag, or move, icon appears when you drag sketch geometry.
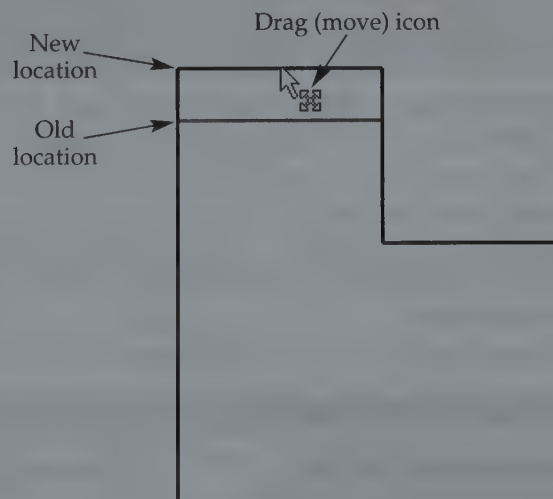
Drag (move) icon

New location

Old location

Another way to analyze constraints is to view degrees of freedom by right-clicking and selecting **View Degrees of Freedom**. Degrees-of-freedom glyphs appear, indicating objects that remain under-constrained. The glyphs represent the directions in which the objects can move. See **Figure 4-21**. To hide a specific degree-of-freedom glyph, right-click on the glyph and deselect **Display Degrees of Freedom**. To hide all degrees-of-freedom glyphs, right-click and select **Hide All Degrees of Freedom**.

Figure 4-21.
Display degrees-of-freedom glyphs to analyze freedom of movement remaining in an under-constrained sketch.

Glyphs associated with a point and multiple degrees of freedom

Glyphs associated with a line and two degrees of freedom

.450

.150

.100

.375

.165

Glyphs associated with a circle and multiple degrees of freedom

Sketch becoming constrained

No degrees-of-freedom glyphs are shown when a sketch is fully constrained
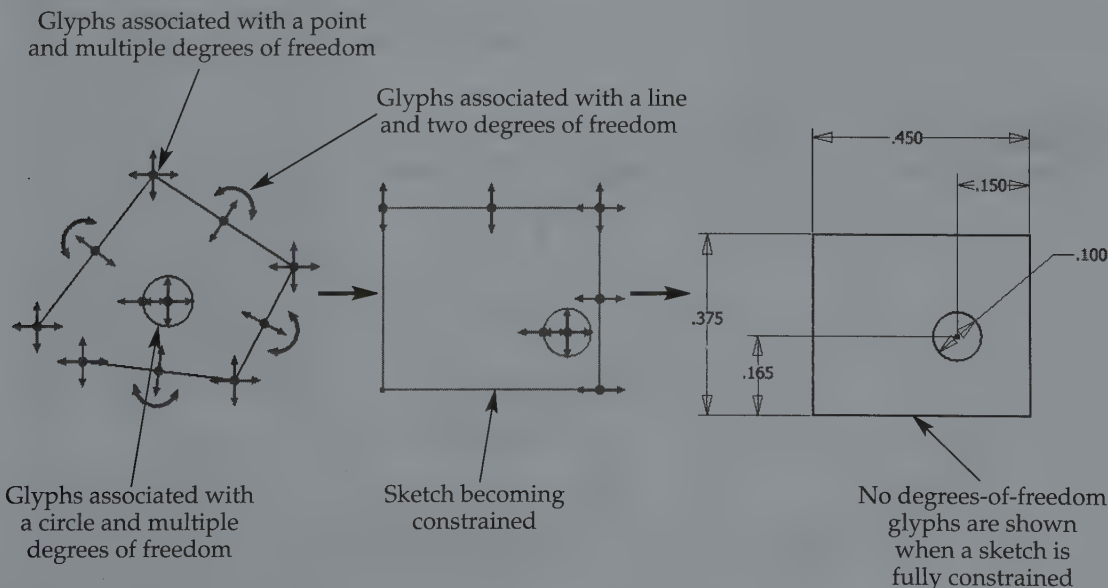
Extending and Trimming Curves
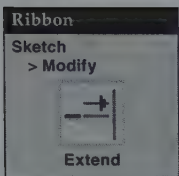
Use the **Extend** tool to increase the length of a curve, such as a line, spline, or arc, to the nearest intersection. Move the cursor over the object to display a preview of the extension. If the preview looks acceptable, pick the object to complete the operation. See **Figure 4-22**. Continue extending objects, or press [Esc], right-click and pick **Done**, or access another tool to exit. Use the **Extend** tool to close gaps to create the closed loop that is usually required to create a solid object.
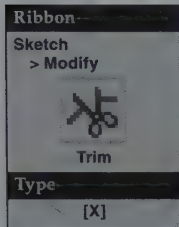
Use the **Trim** tool to remove an unwanted curve that extends past an intersection. Move the cursor over the object to display a preview of the trim. If the preview looks acceptable, pick the object to complete the operation. See **Figure 4-23**. Continue trimming objects, or press [Esc], right-click and pick **Done**, or access another tool to exit.

> **NOTE**
>
> To trim a segment while using the **Extend** tool, or to extend a segment while using the **Trim** tool, hold down the [Shift] key.

Splitting Objects

Access the sketch **Split** tool to separate a single object into two objects at an intersection or apparent intersection. Move the cursor near the intersection or apparent intersection of two objects to highlight the object to be split and display markers at the split locations. See **Figure 4-24**. If the location of the split looks acceptable, pick the

Figure 4-22.
The **Extend** tool extends a selected curve to the nearest intersection or apparent intersection.

Line to be extended

The line is extended after it is picked

A preview of the line extension appears when the cursor is over the line
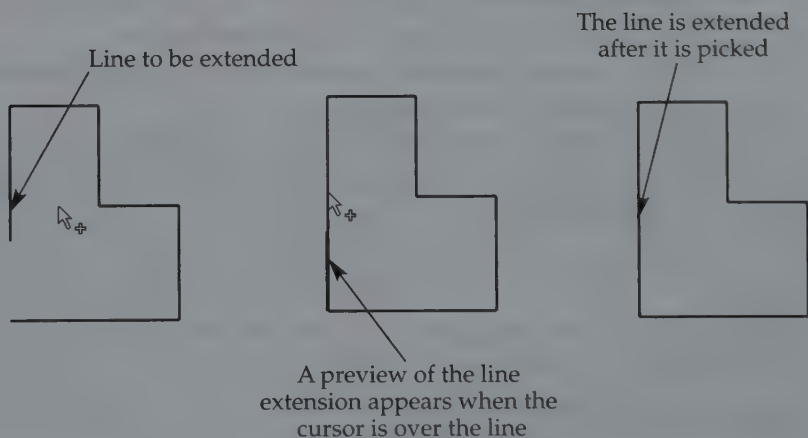
Figure 4-23.
The **Trim** tool trims a selected curve to the nearest intersection or apparent intersection.

Line to be trimmed

A preview of the line trim appears when the cursor is over the line

The line is trimmed after it is picked
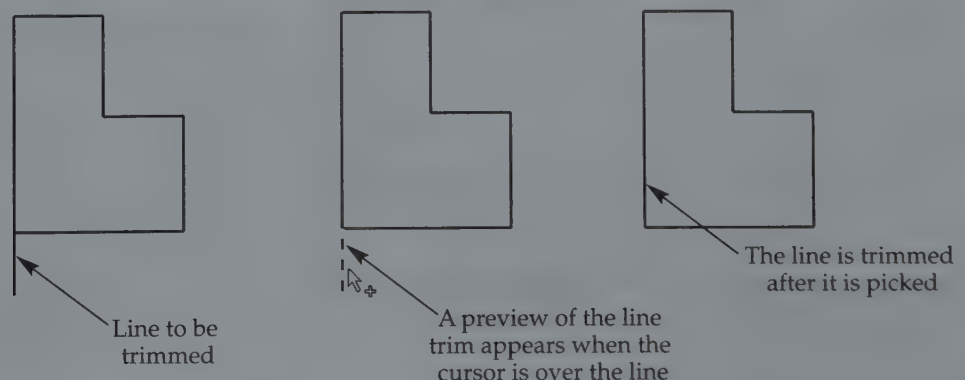
Figure 4-24.
The **Split** tool breaks a selected curve at the intersection or apparent intersection of another curve.

Pick to split at
the intersection

Pick to split at the
apparent intersection
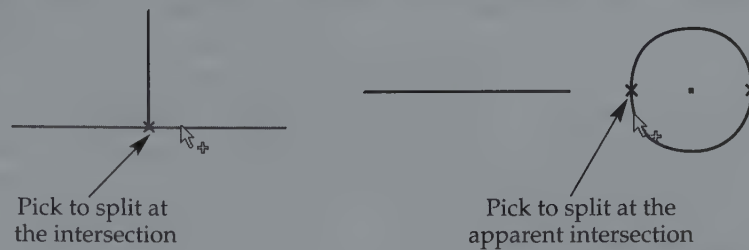
object to create the split. The resulting split segments are individually constrainable. Continue splitting objects, or press [Esc], right-click and pick **Done**, or access another tool to exit.

NOTE

Access the **Trim**, **Extend**, and **Split** tools while trimming, extending, or splitting by right-clicking and selecting the appropriate menu option.

Moving Objects

Access the **Move** tool to move sketch objects, often an entire profile, from one location to another using the **Move** dialog box. See **Figure 4-25**. The **Select** button is active by default, allowing you to pick objects to move. Then right-click and pick **Continue**, or pick the **Base Point** button and select the base point from which to move the objects. Another option is to transition from object selection to base point selection by checking the **Optimize for Single Selection** check box. Pick the **Copy** check box to copy the objects.

Ribbon
Sketch
> Modify
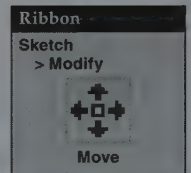
Move

Figure 4-25.
Moving sketched geometry using the **Move** dialog box.

Pick to access the additional
objects shown

Pick to relocate the selected objects

Selected objects

Selected base point
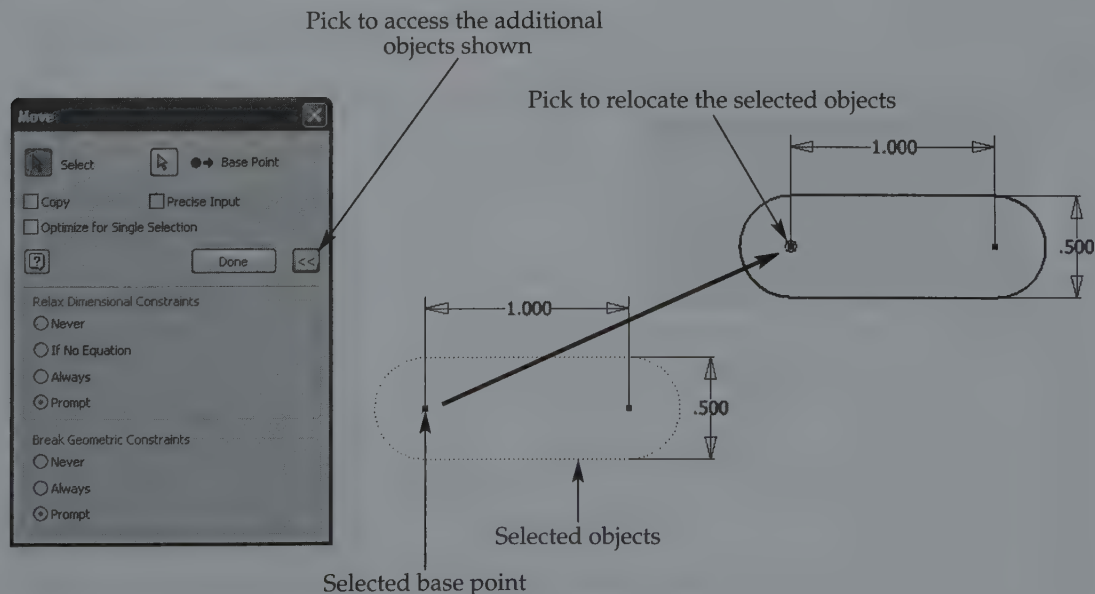
In order to move constrained objects, it is necessary to remove constraints that locate the objects. Pick the **More** button to display options to allowing or prevent removal of constraints. Select a radio button in the **Relax Dimensional Constraints** area to specify whether to allow editing or removal of dimensional constraints. Pick a radio button in the **Break Geometric Constraints** area to specify whether to allow removal of geometric constraints. Both options are set to **Prompt** by default, which displays an alert when it is necessary to relax or break constraints in order to move the selected objects.

Once you specify the options, pick a point to complete the move. You can use the **Inventor Precise Input** toolbar to input coordinates for the relocation point by picking the **Precise Input** check box. Continue moving objects, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

> **NOTE**
>
> The **Optimized for Single Selection** option and similarly named options available with some tools sound advanced, but merely refer to a function that allows you to make faster selections without picking buttons or "continuing through" the dialog box. Inventor cycles through options automatically after you make the appropriate selections.

Copying Objects
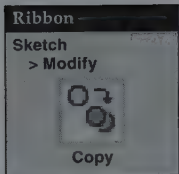
Pick the **Copy** check box in the **Move** dialog box to make a single copy of the selected objects while using the **Move** tool. Access the **Copy** tool to make multiple copies of sketch objects using the **Copy** dialog box. See **Figure 4-26**. The **Select** button is active by default, allowing you to pick objects to copy. Then right-click and pick **Continue**, or pick the **Base Point** button and select the base point from which to copy the objects. The **Optimize for Single Selection** and **Precise Input** check boxes are available and behave the same as those found in the **Move** dialog box. Once you specify the options, pick or enter a relocation point to complete the copy. Continue copying objects, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

Figure 4-26.
Copying sketched objects using the **Copy** dialog box.
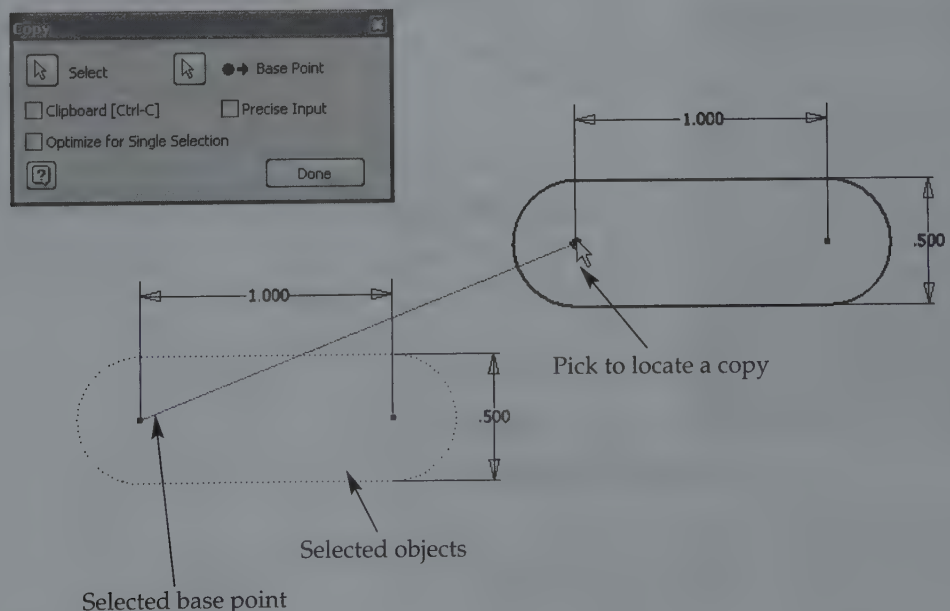
Scaling Objects

Access the **Scale** tool to change the size of objects using the **Scale** dialog box. See **Figure 4-27**. The **Select** button is active by default, allowing you to pick objects to scale. Then right-click and pick **Continue**, or pick the **Base Point** button and select the base point to define the point where the increase or decrease in size occurs. The **Optimize for Single Selection** and **Precise Input** check boxes and constraint disposition options are available and behave the same as those found in the **Move** dialog box.

You can pick a point to specify the new scale, but in most cases, it is appropriate to enter a *scale factor* using the **Scale Factor** text box. For example, to make the selection twice the current size, type 2. Continue scaling objects, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.
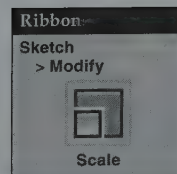
scale factor: The amount of enlargement or reduction.

> **NOTE**
>
> The **Scale** tool allows you to change the size of objects—often an entire profile—proportionately, without adjusting individual dimensions. Dimensions also change to reflect the new size.

Rotating Objects

Access the **Rotate** tool to rotate objects around a point using the **Rotate** dialog box. See **Figure 4-28**. The **Select** button is active by default, allowing you to pick objects to rotate. Then right-click and pick **Continue**, or pick the **Select** button in the **Center Point** area and select a point to define the axis of rotation. Pick the **Copy** check box to copy the objects. The **Optimize for Single Selection** and **Precise Input** check boxes and constraint disposition options are available and behave the same as those found in the **Move** dialog box.

You can pick a point to specify the new rotation angle, but in most cases, it is appropriate to enter a rotation angle using the **Angle** text box. For example, the objects in **Figure 4-28** were originally horizontal, or at 0°, and are rotated to 45°. Continue rotating objects, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.
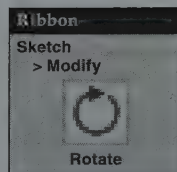
Figure 4-27.
Use the **Scale** tool to proportionately enlarge or reduce the size of sketch objects.
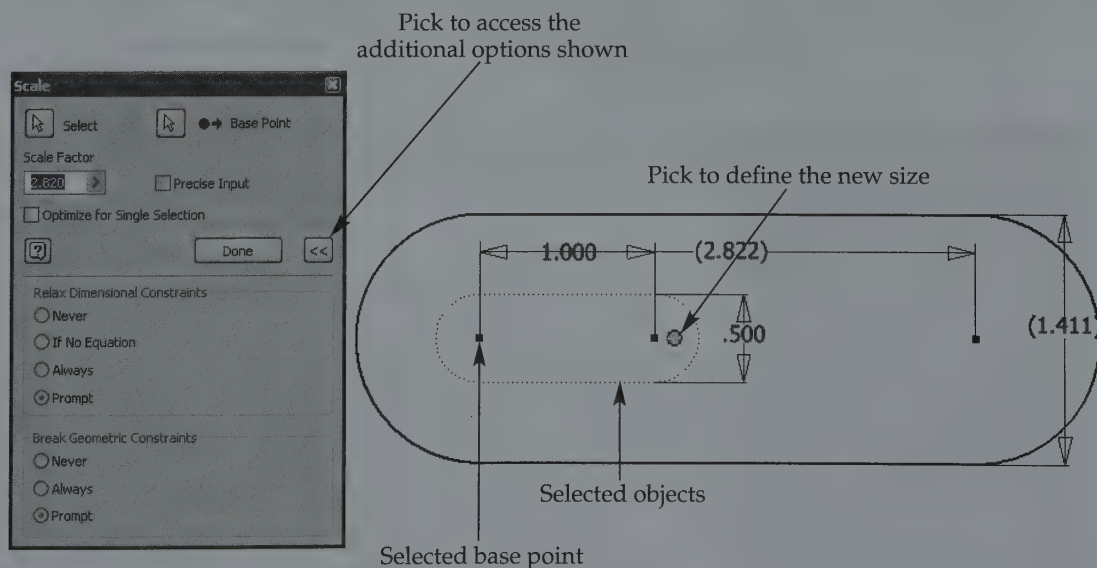
Figure 4-28.
The **Rotate** tool allows you to rotate original sketch geometry and to rotate copies of sketch geometry.

Pick to access the additional options shown

Pick to rotate the selected objects

.500

1.000

1.000

.500

Selected center point   Selected objects
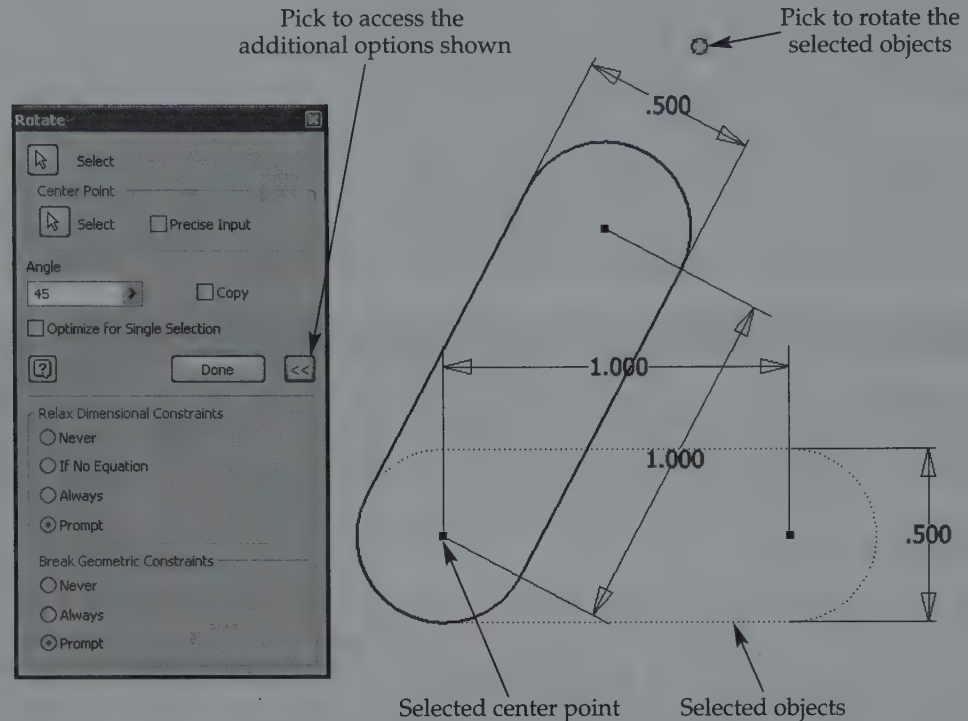
Stretching Objects

Access the **Stretch** tool to stretch objects by modifying certain dimensional constraints using the **Stretch** dialog box. See **Figure 4-29**. The **Select** button is active by default, allowing you to pick objects to stretch. Select *only* the objects to stretch, such as the arc and two lines shown in **Figure 4-29**. If you select a single object or an entire profile, the **Stretch** tool works like the **Move** tool.

Figure 4-29.
The **Stretch** tool allows you to increase or decrease the size of certain objects, such as elongating this boss, or slot, profile.

Pick to access the additional options shown

Pick to stretch the selected objects

1.000   (2.060)

.500   (.500)

Selected objects   Selected base point
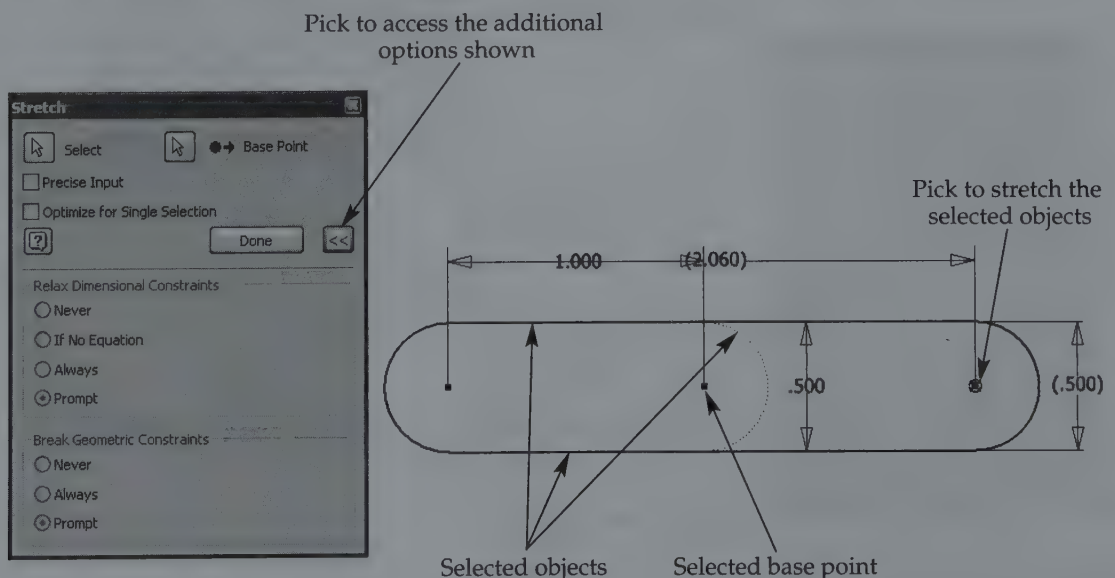
Right-click and pick **Continue**, or pick the **Base Point** select button and choose a point to define the location from which the stretch occurs. The **Optimize for Single Selection** and **Precise Input** check boxes and constraint disposition options are available and behave the same as those found in the **Move** dialog box. Continue stretching objects, or pick the **Done** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

Exercise 4-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-8.

Sketch Parameters

A model *parameter* is automatically created every time you add a dimensional constraint to a sketch. The **Parameters** dialog box, shown in **Figure 4-30**, is an effective tool for controlling model parameters. The **Model Parameters** category contains all standard dimensional constraints. The **Reference Parameters** category is displayed if the model includes driven dimensions.

parameters: Geometric characteristics and dimensions that control the size, shape, and position of model and drawing geometry.

Ribbon
Manage
 > Parameters

fx

Parameters

Figure 4-30.
The **Parameters** dialog box is a good resource to review, edit, and export parameters, such as the sketch dimensional constraints shown.

User parameter created with an equation to control the thickness of the part
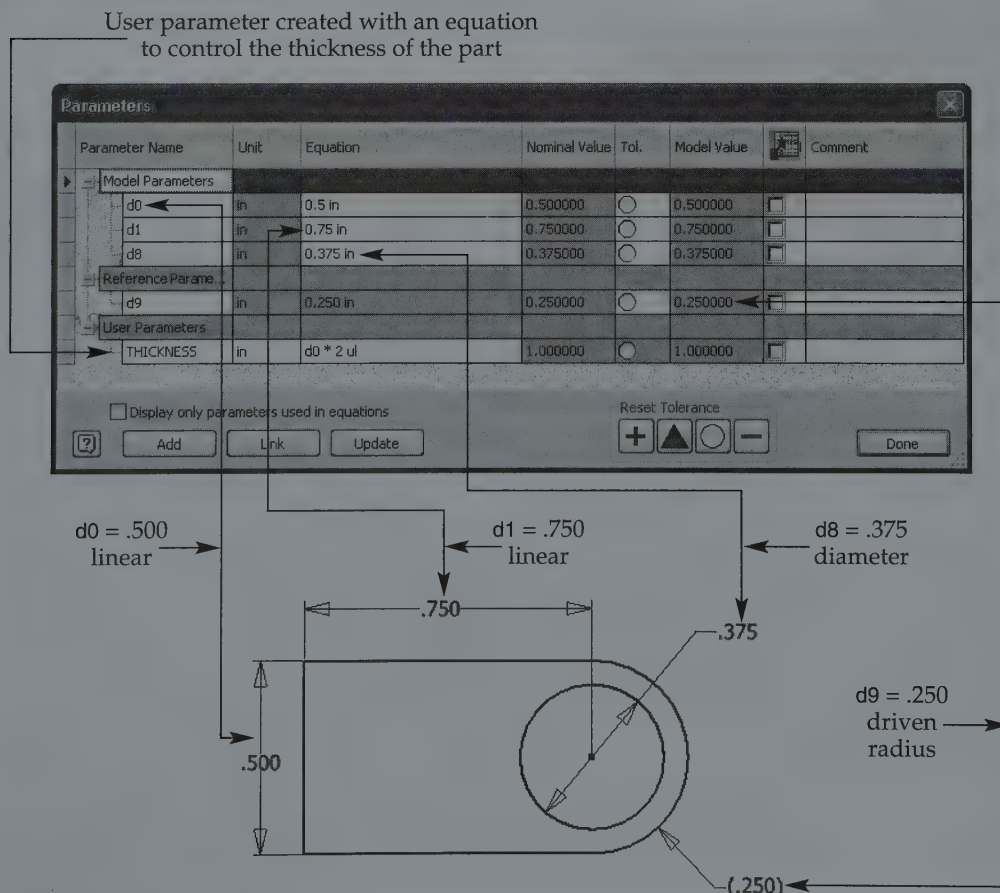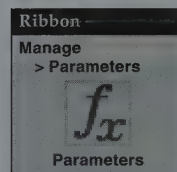
In addition to the automatically specified model and reference parameters, you can specify your own parameters in the **User Parameters** category by selecting the **Add** button. User parameters function just like model parameters in the **Parameters** dialog box. Create parameters in order to access specific parameters throughout the design process. For example, if you know the thickness of a part will always be twice a certain sketch dimensional constraint, create a user parameter similar to the parameter shown in **Figure 4-30** to define the thickness. In addition, when you select the cascading submenu next to a variety of text boxes, the **List Parameters** option is available, which allows you to select a user parameter to add to the text box.

Figure 4-31 describes each column found in the **Parameters** dialog box table. The **Reset Tolerance** area contains **Upper**, **Median**, **Nominal**, and **Lower** buttons that allow you to define or reset the tolerances of all parameters listed in the **Parameters** dialog box to upper, median, nominal, or lower tolerances. Select the **Display only parameters used in equations** check box to show only parameter values used in equations or associated with equations. Pick the **Link** button to link or embed a specified spreadsheet to use for parameters. Press [Enter] or select the **Done** button when you are finished using the **Parameters** dialog box.

> **NOTE**
>
> Use parameters and the **Parameters** dialog box throughout the design process to work with parameters in anything from a basic sketch to the most complex assembly.

> **PROFESSIONAL TIP**
>
> Use the **Region Properties** tool to analyze the properties of sketch regions, or profiles. Select a profile and pick the **Calculate** button to show the region properties. Select an option from the **Dual Units** drop-down list to display properties in alternate units.

Ribbon
Tools
> Measure

Region Properties

Figure 4-31.
The columns in the **Parameters** dialog box allow you to view or manage parameters.

Column	Description
Parameter Name	Initially displays the default parameter name. The first dimensional constraint receives the name d0, the next d1, and so on. Use the text box to change a default name to something more descriptive, such as Length, Width, or Diameter.
Unit	Displays the measurement units applied to dimensional constraints; only editable for user parameters.
Equation	Displays the value or equation specified when you create a dimensional constraint. Use the text box to change a model or user value or equation.
Nominal Value	Shows the *nominal value* of the dimensional constraint.
Tolerance	Provides a drop-down list with nominal, lower, upper, or median tolerance options.
Model Value	Displays the current parameter value or the calculated value of an equation.
Export Parameter	Check to add a dimensional constraint to custom properties for use in applications such as parts lists or a bill of materials.
Comment	Use the text box to enter a comment about the parameter.

nominal value:
The value of a commercial product; intended to be the true drawn size without any specified limits.

Exercise 4-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-9.

Sketch Doctor

Once you create a sketch, the next step in developing a model is to produce a sketched feature. Typically, to create a feature, your sketch should result in a closed loop and be free of defects. Sketch problems or defects are typically the result of unneeded points, a missing coincident constraint, overlapping curves, open loops, or self-intersecting loops. The **Sketch Doctor** can help to resolve sketch problems.

You can also access the **Sketch Doctor** while creating a sketched feature by picking the **Examine Profile Problems** button in the feature tool dialog box. If you access the **Sketch Doctor** from a sketched feature dialog box, you automatically see the **Examine** section of the **Sketch Doctor** dialog box. Otherwise, the **Select** section appears. See **Figure 4-32A**. Pick the **Diagnose Sketch** button to open the **Diagnose Sketch** dialog

Shortcut
Sketch Doctor

Figure 4-32.
Use the **Sketch Doctor** to find problems with a sketch.
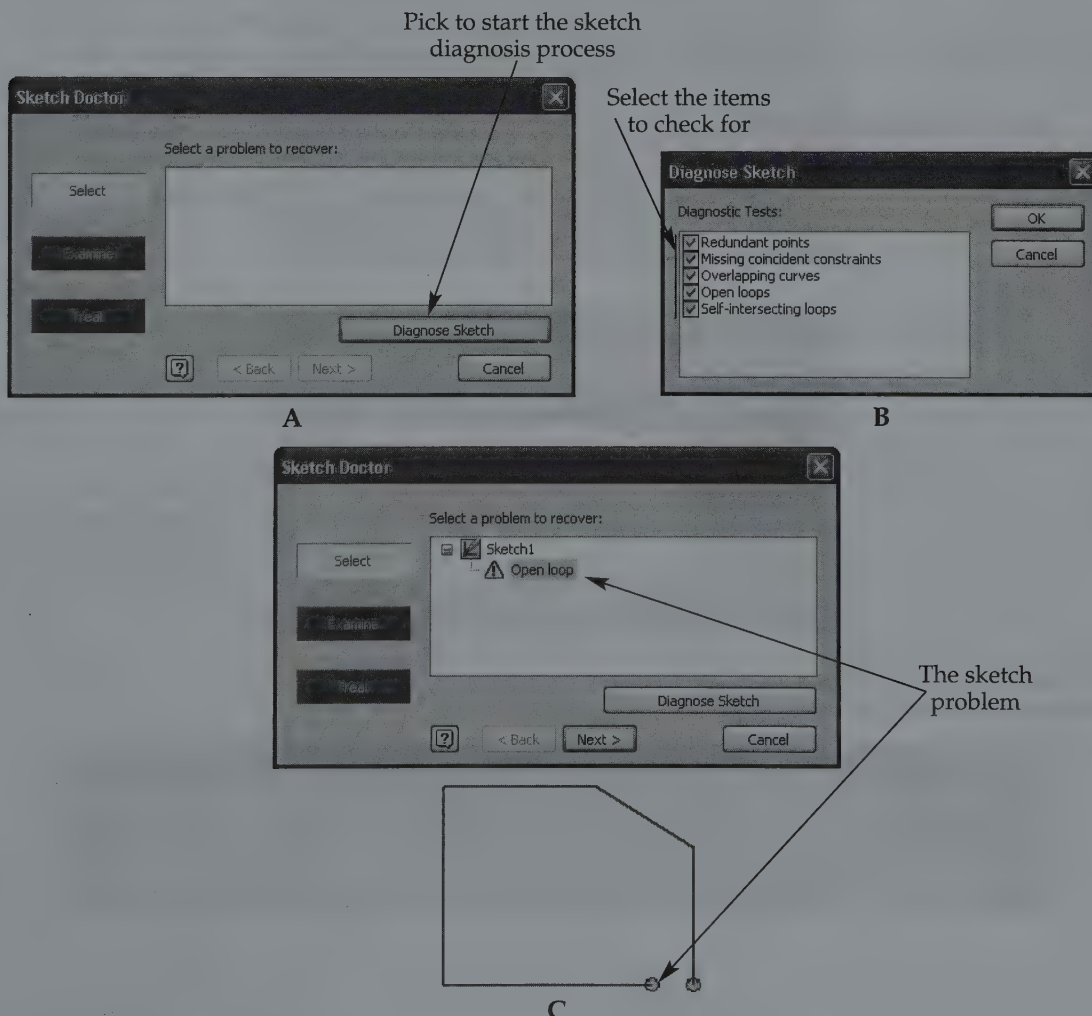
box, where you can choose which diagnostic tests to run. See **Figure 4-32B**. Pick the **OK** button to return to the **Sketch Doctor** with the sketch icons and sketch problems displayed in the **Select a problem to recover:** list box. Notice that the sketch and the sketch problems become highlighted in the graphics window. See **Figure 4-32C**.

Pick the **Next** button to display the **Examine** section of the **Sketch Doctor**. See **Figure 4-33A**. The **Examine** section shows a description of the problem and the diagnosis. You can also highlight or unhighlight the error in the sketch if desired. Pick the **Next** button to access the **Treat** section of the **Sketch Doctor**. See **Figure 4-33B**. The treatment, or problem-solving, options display in the **Select a problem to recover:** list box. Choose the **Edit Sketch** option and pick the **Finish** button. As shown in **Figure 4-33C**, a solution displays in the graphics window. Follow the solution directions to solve the sketch problem.

Exercise 4-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 4-10.

Figure 4-33.
The **Sketch Doctor** presents a diagnosis and a solution.

Problem diagnosis/description:

Open loop found in this profile. Use Close Loop or use Edit Sketch and project geometry, add sketch geometry, and/or drag the loop endpoints to close the loop.

Unhighlight Problem
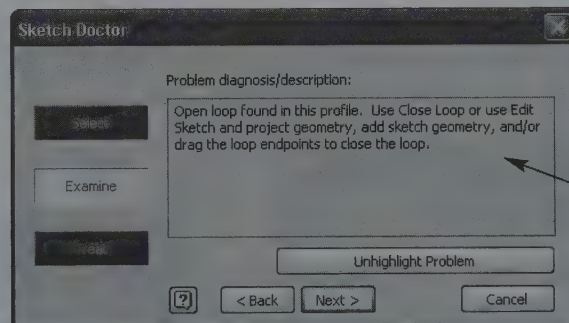
Review the diagnosis

A

Select a treatment:

Close Loop
Edit Sketch
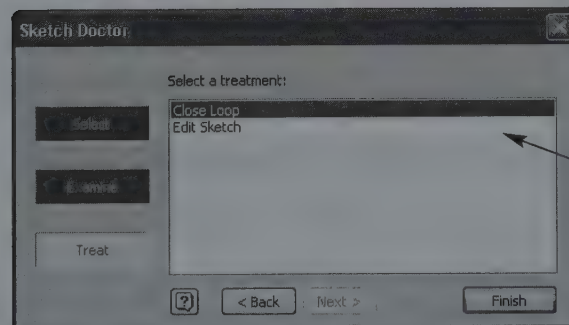
Select the proposed solution or choose to edit the sketch directly

B

This message displays if you select the proposed solution
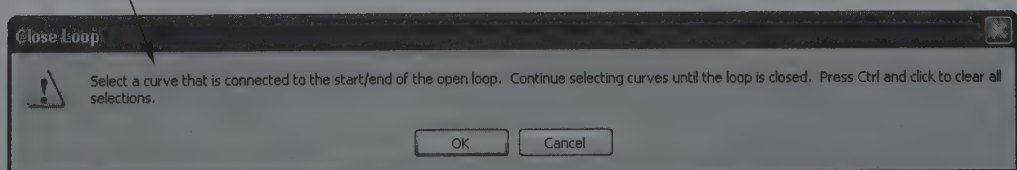
Select a curve that is connected to the start/end of the open loop. Continue selecting curves until the loop is closed. Press Ctrl and click to clear all selections.

OK   Cancel

C

Chapter Test

1. Briefly describe what happens to the first and second selected objects when you are using constraint tools that require you to pick two objects.
2. What are constraint glyphs?
3. How do you recognize which objects and constraints are associated with a glyph?
4. What is the function of dimensional constraints in a parametric sketch?
5. What types of characteristics can you define using a linear dimensional constraint?
6. What is an aligned dimensional constraint?
7. What is the radius of a circle that has a 24-mm diameter?
8. What is a driven dimensional constraint, and what visual cue differentiates it from other dimensional constraints?
9. What is a sketch pattern?
10. What is the axis of rotation for a circular pattern?
11. Describe the basic function of the **Offset** tool.
12. What is the purpose of the **Inventor Precise Input** toolbar?
13. Explain two methods of determining whether a sketch is adequately constrained.
14. How can you trim a line while the **Extend** tool is active?
15. What is the difference between the **Trim** tool and the **Split** tool?
16. Briefly describe two ways to copy selected geometry.
17. What happens if you access the **Stretch** tool and then pick all of the lines of a rectangle to stretch?
18. Define *parameters*, and explain when model parameters are created.
19. When might you want to create a user parameter?
20. Briefly describe the function of the **Sketch Doctor**.

Problems

1–7 Instructions:

- *Open the specified Chapter 3 part file, and save the file using the name given.*
- *Reopen the existing sketch to make changes.*
- *Fully constrain the sketch. Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure. Add the dimensional constraints shown.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

▼ Basic

1. **File:** P3-1.ipt
   Save as: P4-1.ipt
   Title: PIN
   Specific instructions: Add the diameter dimensional constraint to fully constrain the sketch.

4.75
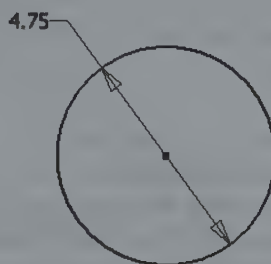
▼ Basic

2. **File:** P3-2.ipt
   Save as: P4-2.ipt
   Title: SWIVEL
   Specific instructions: Add the diameter dimensional constraint to fully constrain the sketch.

19
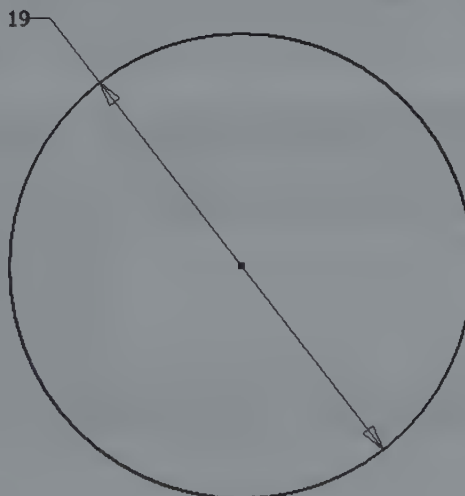
3. **File:** P3-3.ipt
   Save as: P4-3.ipt
   Title: SCREW
   Specific instructions: Add the diameter dimensional constraint to fully constrain the sketch.
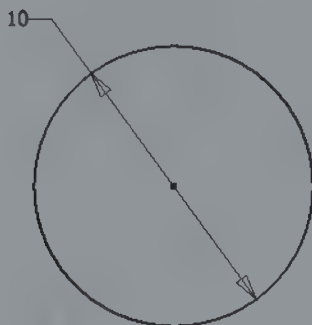
4. **File:** P3-4.ipt
   Save as: P4-4.ipt
   Title: SUPPORT BRACKET
   Specific instructions: Use **Equal** constraints to make both .250 circles the same diameter. Use **Equal** constraints to make both .750 arcs the same radius. The arcs are tangent to the circles.
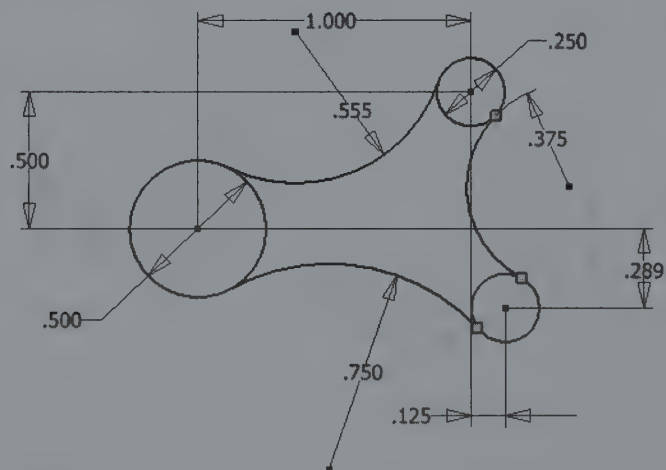
5. **File:** P3-5.ipt
   Save as: P4-5.ipt
   Title: SUPPORT BLOCK

Collinear
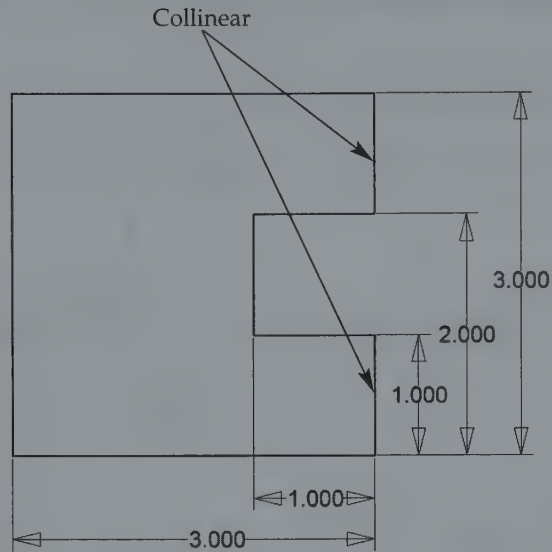
6. **File:** P3-6.ipt
   Save as: P4-6.ipt
   Title: BODY

8–10 Instructions:

- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Do not assign material and color properties at this time.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

8. **Title:** HANDLE <<right "tag end" of the centerline should be coincident with the vertical line as shown on the left>>
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-005-01
   Project: HANDLE
   Sketch plane: XY
   Save as: P4-8.ipt
   Specific instructions: Sketch a horizontal centerline that has a left endpoint coincident to the projected center point. Sketch the largest arcs and the horizontal line above the centerline first. Then mirror the objects over the line of symmetry. Close the shape with two vertical lines, and offset the entire loop.
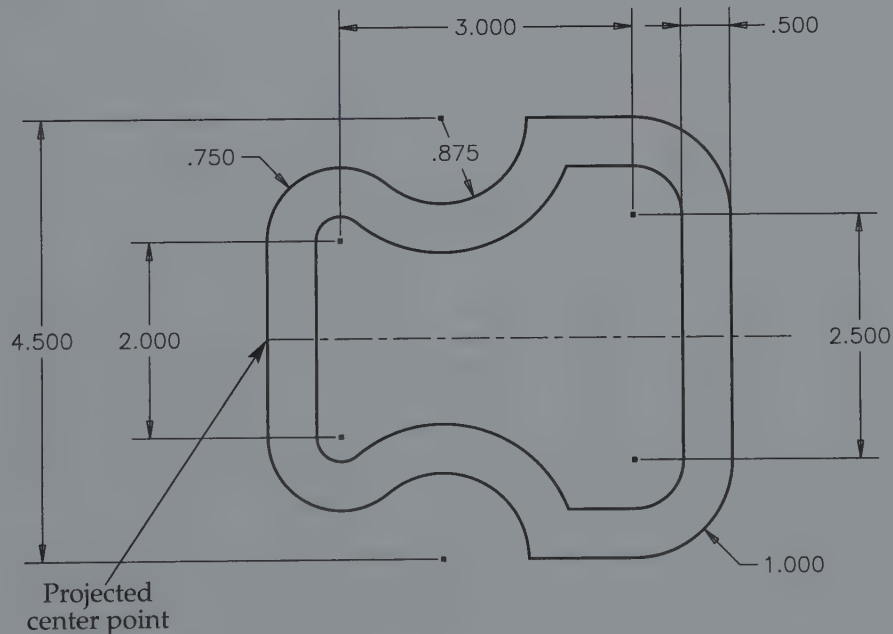
9. **Title:** STOP
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-006-01
   Project: HANDSET
   Sketch plane: XY
   Save as: P4-9.ipt
   Specific instructions: Position the sketch at the projected center point by inferring a coincident constraint as shown.

R1.000

.050

.925

.110

.200

.185

75°

.080

Projected
center point

.204
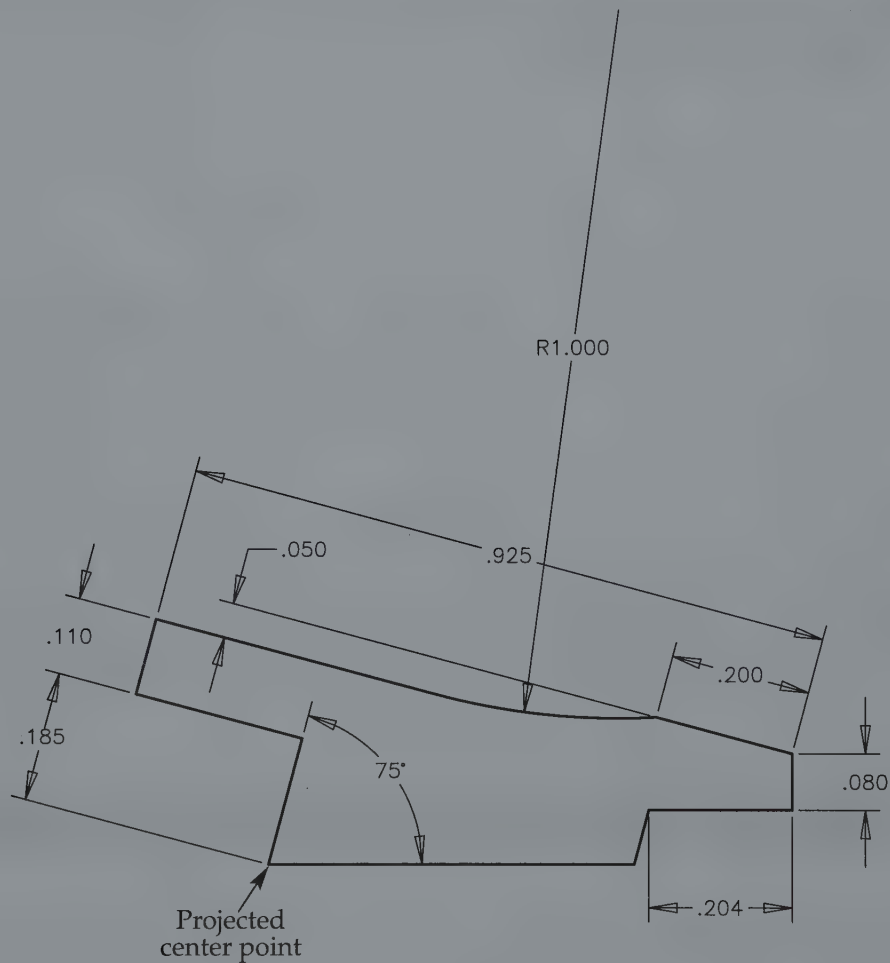
10. **Title:** FLAT SPRING
    Units: Inch
    Template: Part-IN.ipt
    Part Number: IAA-007-01
    Project: EXTENDER
    Sketch plane: XY
    Save as: P4-10.ipt
    Specific instructions: Position the sketch at the projected center point by infer-ring a coincident constraint as shown. All objects are arcs, except for top hori-zontal line. Multiple tangent constraints are required.
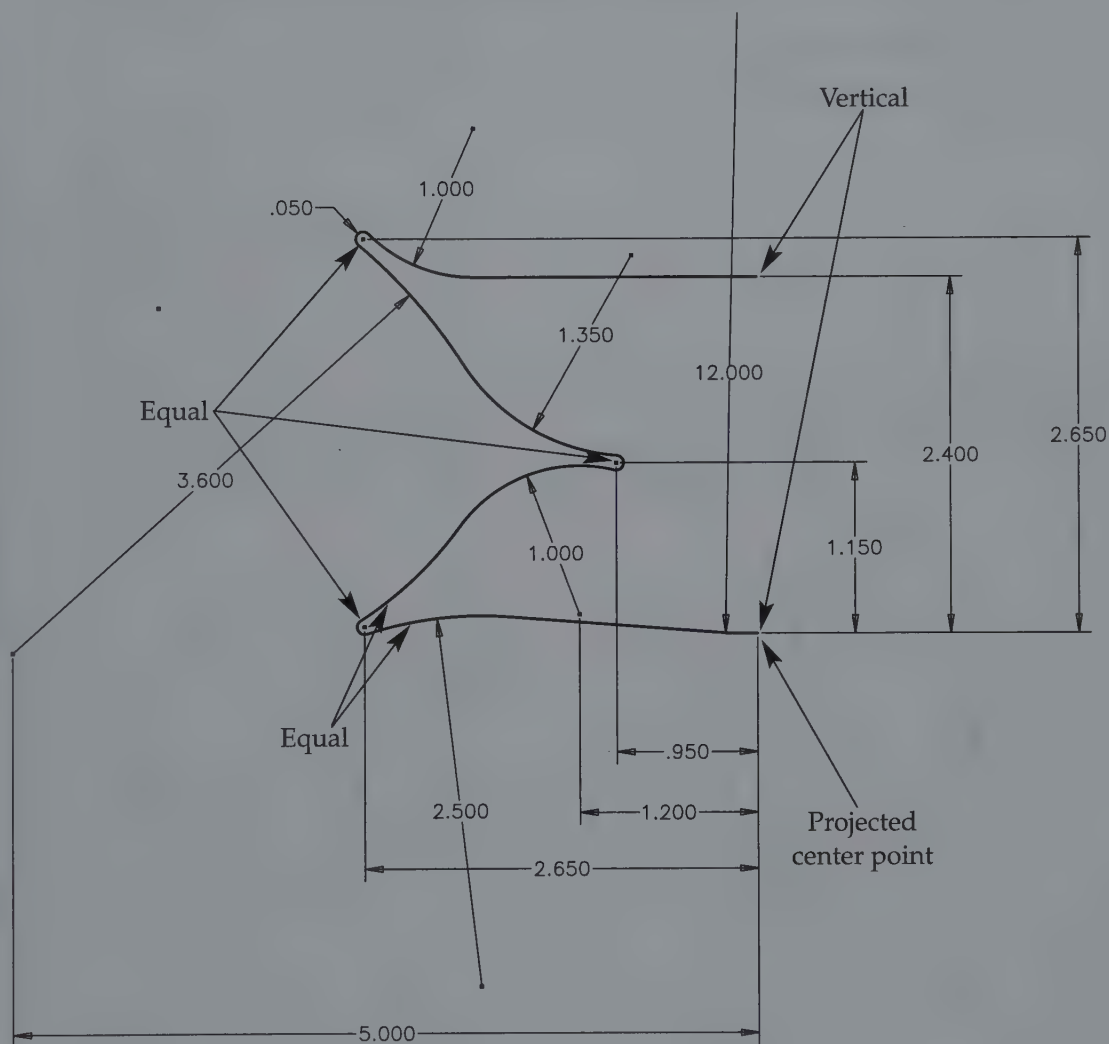
Extrusions, Revolutions, and Coils

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create extruded and revolved base features.
✓ Add features to a base feature.
✓ Develop coils.
✓ Edit features and feature properties.

Usually, the first part model feature, or *base feature,* is a *sketched feature.* Once you build a base feature, you can add other features, including more sketched features, to complete the part design. This chapter explains how to create common extruded and revolved sketched features. You will also learn to create coils.

base feature: The initial model feature, on which all others are based.

sketched feature: A feature built from a sketch. Sketched features include extrusions, revolutions, sweeps, lofts, and coils.

Extruded Base Features

Access the **Extrude** tool to create an *extrusion* using the **Extrude** dialog box. See **Figure 5-1**. Fewer options are available when creating a base feature than when building additional features. If a sketch includes one profile, the profile becomes selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile(s) to extrude.

You have the option of extruding a closed profile as a solid or *surface* by picking the appropriate button in the **Output** area. You can only extrude an open profile as a surface when creating a base feature. **Figure 5-2** shows examples of surface extrusions. When you are creating a solid base feature, the only possible operation is to form a new solid body, as evident by the selected **New Solid** button. Additional operations are possible when adding features.

The **Extents** area contains options for specifying the size and boundaries of the extrusion. The **Distance** option is the only usable **Extents** parameter for creating a base feature in a model that does not include any work planes offset from the sketch plane. Use the **Distance** option to end an extrusion a specified distance from the sketch plane. Use the **Distance** text box to enter a distance, select a value from the list, choose a model parameter, or measure a distance.

Ribbon

Model
> Create

Extrude

Type

[E]

extrusion: A feature with a sketch profile that extends (extrudes) along a linear path.

surface: A volumeless shape primarily used for construction purposes, allowing you to generate more advanced models.

Figure 5-1.
The **Extrude** dialog box allows you to select a sketch profile to extrude and specify extrusion parameters and settings. The **Shape** tab controls primary extrusion shape characteristics. The preview provides a good idea of how the extrusion will appear and helps you select appropriate options.
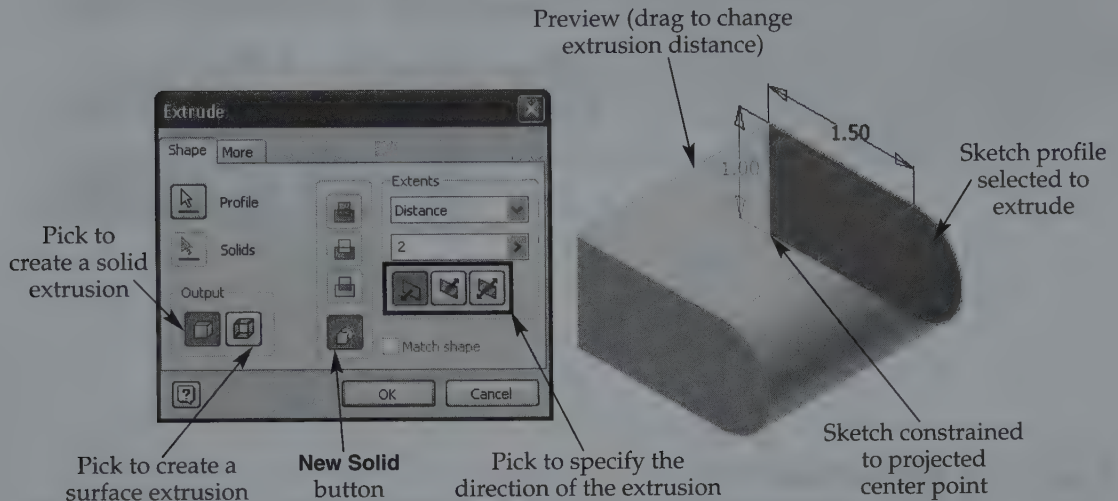
Figure 5-2.
Examples of volumeless surface extrusions. Surfaces appear semitransparent by default and are listed as surfaces in the browser. Surfaces created using Inventor are primarily for construction purposes.
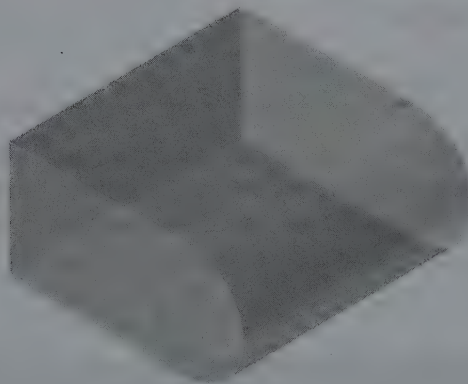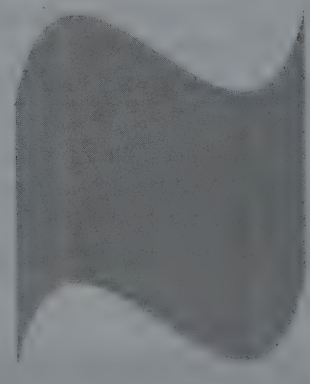
Closed-Loop Profile   Open-Loop Profile

PROFESSIONAL TIP

Enter an extrusion distance using a parameter created in the **User Parameters** section of the **Parameters** dialog box by selecting the **List Parameters** option of the **Distance** cascading submenu and then selecting the appropriate parameter.

Define the extrusion direction by selecting the **Positive**, **Negative**, or **Midplane** direction button. See **Figure 5-3**. You have the option of specifying the extrusion distance and direction by dragging the preview to or from the sketch plane. This is primarily a preview and design function. For most applications, enter an exact value and choose the appropriate direction.

The **More** tab, shown in **Figure 5-4**, controls additional extrusion characteristics. Use the **Taper** text box to taper the extrusion. Specifying a taper angle allows you to apply a slight draft angle or create a very different feature. The options in the **Alternative Solution** area are unavailable for base extrusions. Pick the **OK** button to create the extruded feature.

Figure 5-3.
Choose an extrusion direction appropriate for the application.
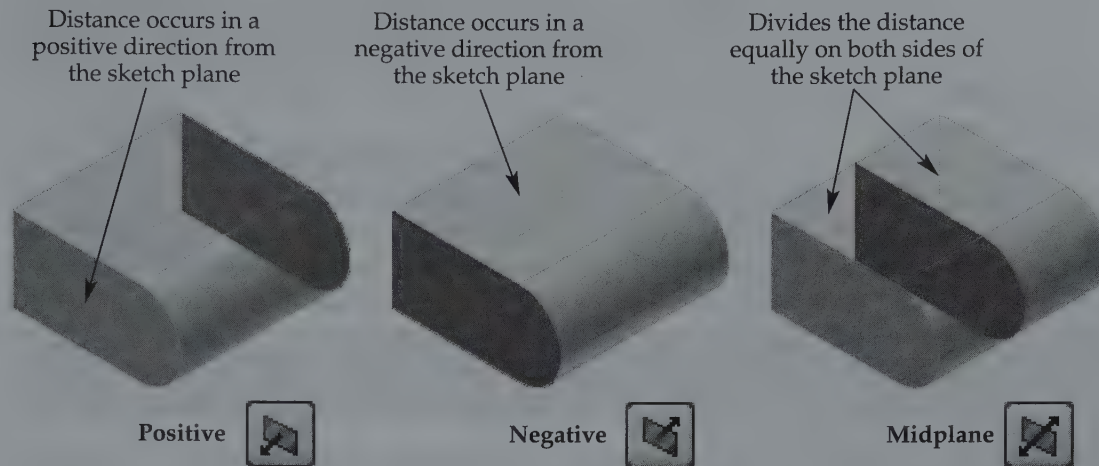
Distance occurs in a positive direction from the sketch plane

Distance occurs in a negative direction from the sketch plane

Divides the distance equally on both sides of the sketch plane

Positive

Negative

Midplane

Figure 5-4.
The **More** tab of the **Extrude** dialog box provides advanced extrusion options and a method for tapering the extrusion. Notice the significant impact direction can have when tapering an extrusion.

Specify an angle other than 0 to taper the extrusion from the sketch plane

Extrude Extrusion2

Shape | More

Alternate Solution

Minimum Solution

Taper

15 deg

OK   Cancel

Negative Direction

Midplane Direction
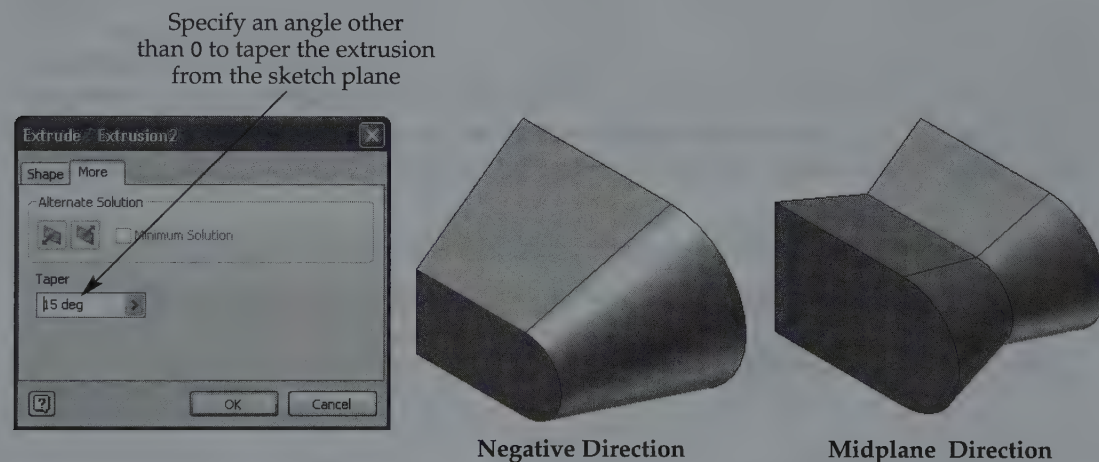
PROFESSIONAL TIP

Once you complete a sketch, access sketch feature tools such as **Extrude** from the sketch environment by right-clicking and selecting a tool from the **Create Feature** cascading submenu.

Exercise 5-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-1.

Basic Model Viewing

A 3D model is created in 3D space and often requires viewing in a 3D orientation. As you progress through the design process, you will adjust the orientation to display specific areas of the model. For example, when you add a sketched feature to a base extrusion, you may have to rotate the base extrusion into a position that allows you to view and select a specific face on which to sketch.
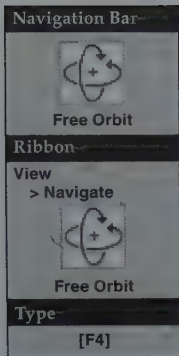
Access the **Orbit** tool to view a model from any angle. The **Orbit** tool is set to **Free Orbit** mode by default. **Figure 5-5** shows the behavior of the **Orbit** tool. Hold down the left mouse button and drag the orbit icon to orbit freely. To orbit around the vertical axis line, drag one of the horizontal lines left or right. To orbit around the horizontal axis line, drag one of the vertical lines up or down. To orbit around the horizontal axis point, drag near an outside quadrant. You can also use the **Orbit** tool to pan the display. Instead of holding down the left mouse button to drag, pick a point to pan the display away from the selection. Press [Esc], pick outside the rotation area, right-click and pick **Done**, or access another tool to exit.

You will learn many additional model viewing tools and options. For now, the **Free Orbit** and **View Face** tools and the basic zooming and panning techniques described in previous chapters should allow you to view models as needed.

Reusing a Sketch

consumes: Uses up in the process of forming a feature.

share: Reactivate a consumed sketch to use for other feature development.

A sketched feature *consumes* a sketch making the sketch unavailable to create additional features. However, you can *share* a consumed sketch to create independent features. To share a sketch, expand the feature consuming the sketch in the browser to show the sketch, right-click on the sketch, and select **Share Sketch**. A new instance of the sketch appears in the browser and becomes active. Use all or part of the sketch to create another feature.

Figure 5-5.
The position of the cursor determines the method of rotation when you use the **Orbit** tool.
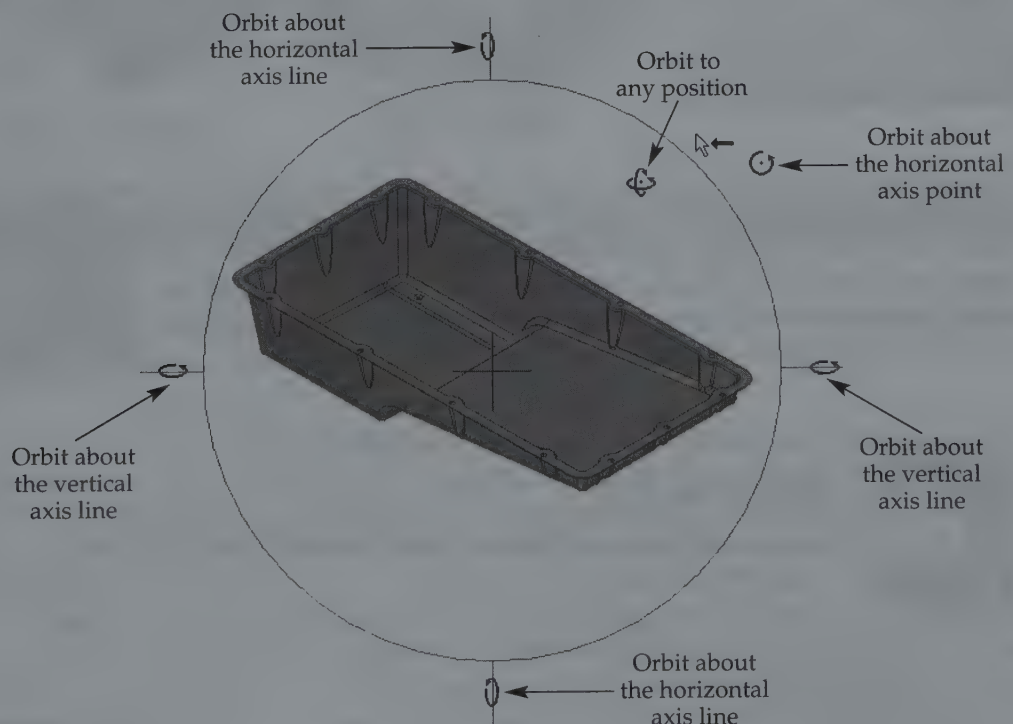
You can also reuse any visible sketch. To make a sketch visible, expand the feature consuming the sketch in the browser to show the sketch, right-click on the sketch, and select **Visibility**. Use the visible sketch to build an appropriate feature. A visible sketch used to create an additional feature is automatically shared.
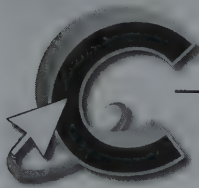
Exercise 5-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-2.

Revolved Base Features

Access the **Revolve** tool to create a *revolved feature*, or *revolution*, using the **Revolve** dialog box. See **Figure 5-6**. Fewer options are available for creating a base feature than for building additional features. If a sketch includes one profile and a line assigned the centerline format, the profile and axis select automatically. If a sketch includes one profile, but not a line assigned the centerline format, the **Axis** button is active, allowing you to pick the axis. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile(s) to revolve. You must then select the **Axis** button to pick the axis.

The axis defines the centerline of revolution. You must consider the location of the axis when you develop the sketch. An axis can be any line in the sketch, an axis in the **Origin** folder of the browser, or a work axis. Inventor recognizes lines in the the sketch centerline format and the construction and centerline format as axes, which can auto-mate selection as shown. The sketch in **Figure 5-6**, for example, uses an axis projected

revolved feature (revolution): A feature with a sketch profile that rotates (revolves) around an axis.

Ribbon

Model > Create

Revolve

Type

[R]
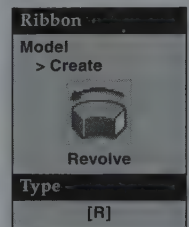
Figure 5-6.
The **Revolve** dialog box allows you to select a sketch profile to revolve and specify revolution parameters and settings. The **Shape** tab controls primary revolution shape characteristics. The preview provides a good idea of how the revolution will appear and helps you select appropriate options.

Preview

Sketch includes projected axis (Y axis) assigned a centerline format

Sketch profile selected to revolve (sketched on XY plane)

Pick to create a solid extrusion

Pick to create a surface revolution

New Solid button

Linear diameter dimensional constraints form by default when you pick a centerline

Sketch constrained to projected center point and Y axis
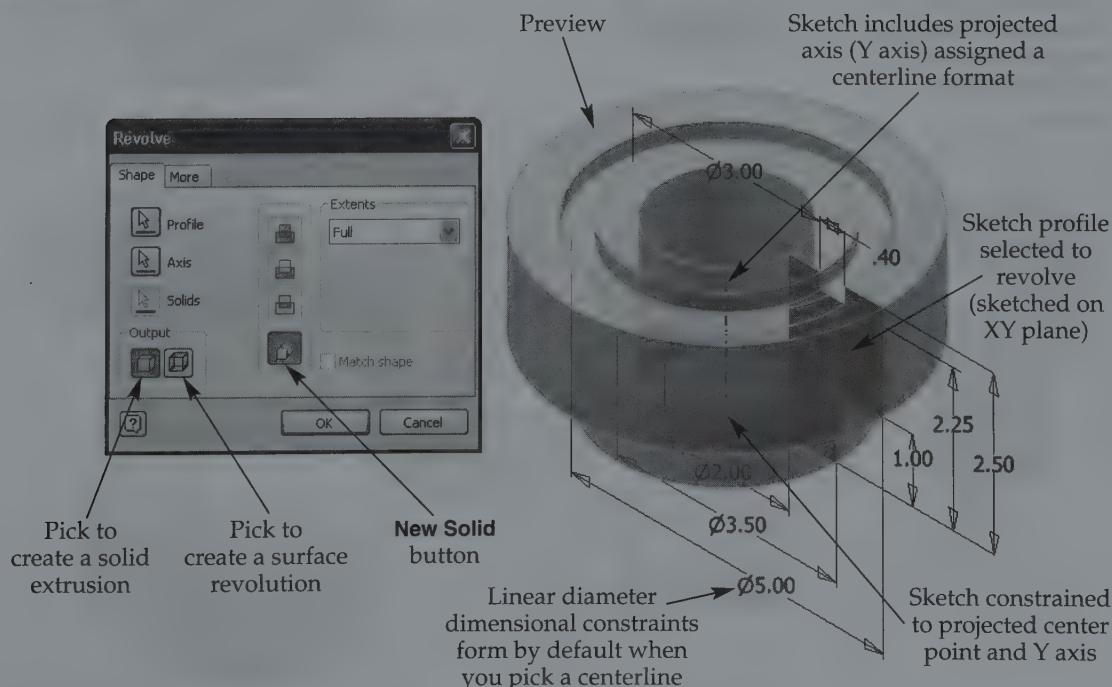
from the **Origin** folder of the browser, converted to a centerline format. The centerline format provides an additional advantage by allowing you to create linear diameter dimensional constraints by default, which is often appropriate for revolved features.

You have the option of revolving a closed profile as a solid or surface by picking the appropriate button in the **Output** area. You can only revolve an open profile as a volumeless surface when creating a base feature. When creating a solid base feature, the only possible operation is to form a new solid body, as evidenced by the selected **New Solid** button. Additional operations are possible for adding features.

The **Extents** area contains options for specifying the angle, or amount, of revolution. The **Full** option is active by default and revolves the profile 360°, or completely around the axis, as shown in **Figure 5-6**. To make an incomplete revolution, as shown in **Figure 5-7**, pick the **Angle** option from the drop-down list and specify a revolution angle between 0° and 360° in the **Angle** text box.

Define the direction of rotation by selecting the **Positive**, **Negative**, or **Midplane** direction button. When using the **Angle** option, you can specify the revolution angle and direction by dragging the preview to or from the sketch plane. This is primarily a preview and design function. For most applications, enter an exact value and choose the appropriate direction. The **More** tab controls additional revolution characteristics that are unavailable for base revolutions. Pick the **OK** button to create the revolved feature.

Figure 5-7.
Select the **Angle** option of the **Extents** area to specify an incomplete revolution angle and direction. Notice the examples of different angles shown.

Revolution occurs clockwise from the sketch plane

Revolution occurs counterclockwise from the sketch plane

Divides the revolution equally on both sides of the sketch plane
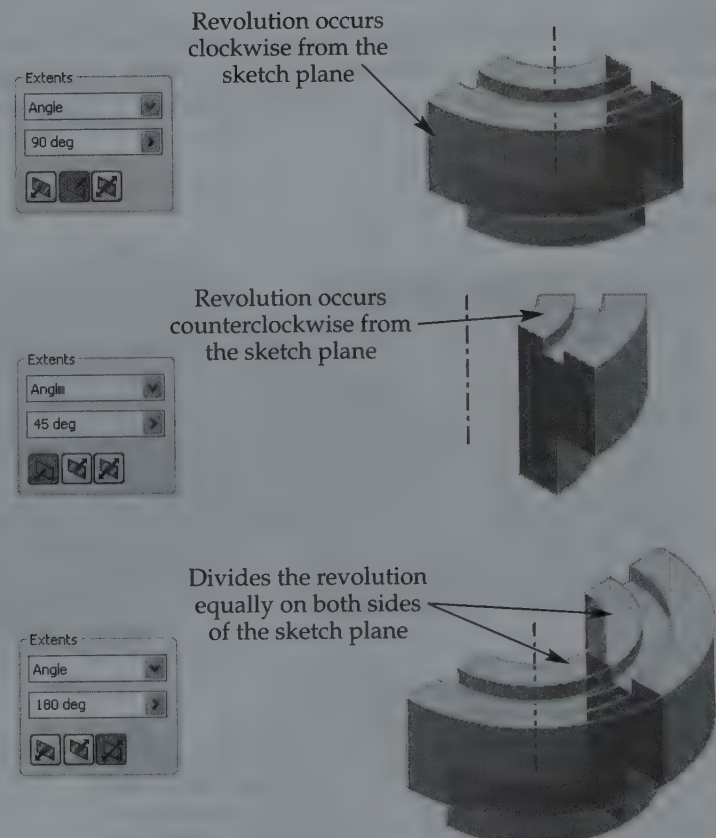
Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-3.

Adding Features

Once you create a base feature, you are ready to continue part development by adding features. The next feature can be a sketched, placed, catalog, pattern, or work feature. In general, additional sketched features, and in some cases work features, reference a sketch. You *place* all other features on existing feature geometry without using a sketch.

Adding 2D Sketches

Use the **2D Sketch** tool to place an additional sketch. The process is the same as beginning a base feature sketch. You can access the tool before or after specifying the sketch plane. Several options are available for adding a sketch to a model that already contains a feature. One common technique is to open a sketch on the face of an existing feature. See **Figure 5-8**. Another common option is to open a sketch on one of the default work planes in the **Origin** folder of the browser. This is the same as picking a plane to begin a base feature sketch. See **Figure 5-9**.

Ribbon
Model > Sketch
Create 2D Sketch
Shortcut
New Sketch
Type
[S]
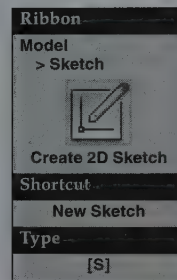
PROFESSIONAL TIP

To sketch on a surface that is hidden by existing features, right-click and select **Slice Graphics** to temporarily section the part at the sketch plane, revealing the sketch geometry.

Shortcut
Slice Graphics
Type
[F7]
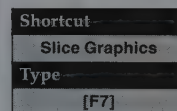
Figure 5-8.
Examples of sketching on feature faces to add extrusions required to develop a socket-head cap screw. The sketches in this example are constrained to the projected center of the screw head to form the shaft and socket.

Existing revolution — Pick a planar face to sketch — .190 — **Shaft Sketch**

Join extrusion — **Shaft Feature**

Pick a planar face to sketch — .126 — **Socket Sketch**

Cut extrusion — **Socket Feature**
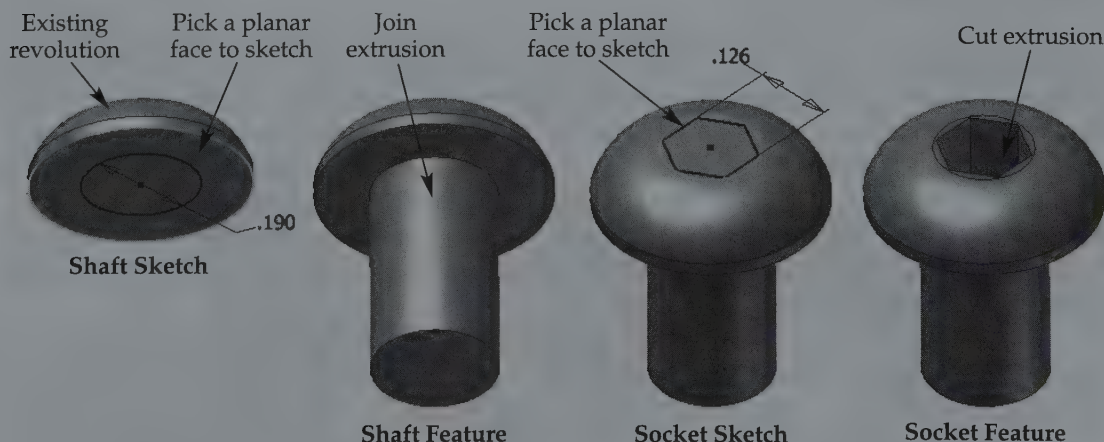
Figure 5-9.
Examples of
sketching on planes
available from
the **Origin** folder
of the browser to
add extrusions
required to develop
the ball for a ball
valve. The sketches
in this example
are constrained
to the projected
circumference of the
ball.

.850

XY plane

Base Feature Sketch

Revolved Base Feature

Sketch
profiles

580

XY plane
(Feature semi-
transparent)

Second Feature Sketch

Cut Extrusion

Sketch
profile

.360

YZ plane
(Feature semi-
transparent)

Third Feature Sketch

Cut Extrusion
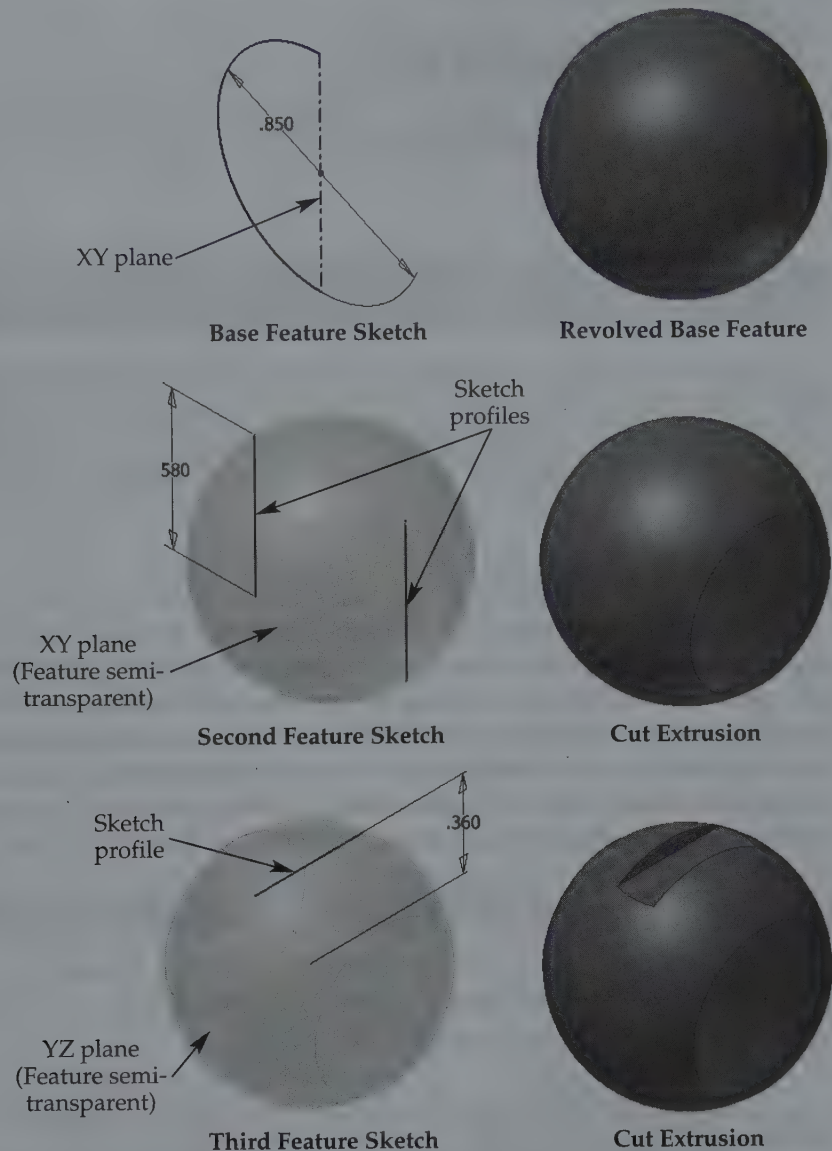
Constraining to Features

Several tools and options are available for reusing existing model geometry to construct additional sketches. One of the easiest techniques is to select feature points and edges when applying geometric and dimensional constraints. Use the same procedures as you would to select sketch geometry. The points and edges you constrain to are automatically projected onto the sketch plane. See **Figure 5-10**. To reference existing feature parameters when adding dimensional constraints, pick the **Show Dimension** option from the **Edit Dimension** dialog box menu, and then select a feature to show dimensional constraints. Pick a value to copy the associated parameter to the **Edit Dimension** dialog box.

You can also use the **Project Geometry** tool to project specific feature points and edges onto the sketch plane. This is the same tool you use to project the default work features, available from the **Origin** folder of the browser, onto a base feature sketch. Use the **Project Geometry** tool to project any available construction geometry. You can project and reuse the items in the **Origin** folder of the browser as many times as needed.

Ribbon
Sketch
> Draw
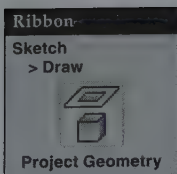
Project Geometry

Figure 5-10.
Adding geometric
constraints and
dimensional
constraints from the
sketch to existing
features without
first projecting
feature objects onto
the sketch plane.

Dimensional constraints
reference feature angles

Parallel geometric
constraint references
feature edge

fx:1.00   fx:1.00

1.50
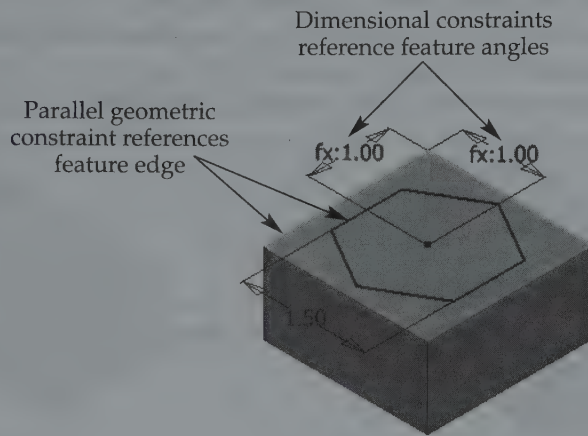
Using Autoproject Tools

Tools are also available that project edges automatically onto a new sketch plane. The **Sketch** tab of the **Application Options** dialog box includes two automatic projection tools. Pick the **Autoproject edges during curve creation** check box to project a feature edge onto the sketch plane while using sketch tools. To use the **Autoproject edges during curve creation** tool, access a sketch tool, such as **Line**, **Spline**, **Circle**, or **Arc**, and then move the cursor over the edge to project. The edge changes color to signify projection.

Edges projected using the **Autoproject edges during curve creation** tool initially appear for reference. They become sketch geometry only when you constrain to the projection. If you do not actually pick a location on a projected edge while sketching to apply a coincident constraint, the projection is removed when you exit the sketch tool and is unavailable for feature development. See **Figure 5-11**.

> **NOTE**
>
> Toggle the **Autoproject edges during curve creation** tool on and off while using a sketch tool by right-clicking and selecting or deselecting **Autoproject**.

Figure 5-11.
A—Autoprojecting a feature edge while using the **Rectangle** tool. B—Autoprojecting a second feature edge and sketching a rectangle by inferring coincident constraints on projected edges. C—Extruding the rectangle to the closest face.

Hover to project edge

Second
reference edge

.625

Projected
reference
edge

Sketch
plane

A   B   C
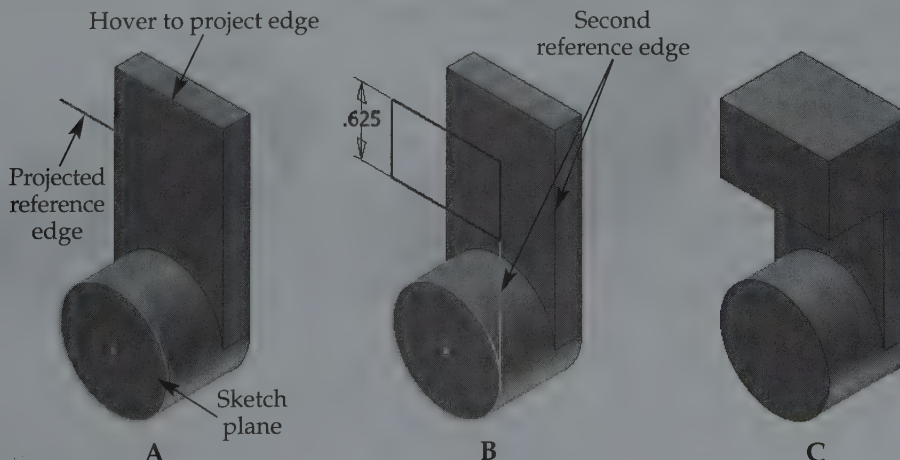
The **Autoproject edges for sketch creation and edit** check box, which is selected by default, projects and constrains all face edges onto the sketch plane when you open a new sketch. See **Figure 5-12**. Edges change color to signify projection. Edges projected using the **Autoproject edges for sketch creation and edit** tool become sketch geometry. Use this technique to produce a new sketch profile almost instantly based on the outline of a feature face.
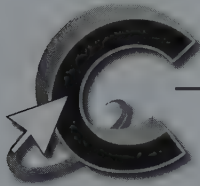
Exercise 5-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-4.

Adding Extrusions

The **Extrude** dialog box provides additional operation, extents, and solution options for adding an extrusion to a model with existing features. Pick the **Join** button to add material to the solid. Joining is essentially the same operation applied when you create a base feature. Select the **Cut** button to use the sketch profile to remove material from the solid. Pick the **Intersect** button to retain the intersecting portion of the new extrusion and existing solid. Disregard the **New solid** button for now. **Figure 5-13** shows the application of different extrusion operations using the same sketch profile.

The **Distance** extents option remains available, but you can now pick **Measure** from the **Distance** list to measure points and curves on a feature. Pick **Show Dimensions** from the **Distance** list and select an existing feature to see dimensional constraints associated with the feature. You can then select a dimensional constraint value, or parameter, to copy to the **Distance** text box.

Pick the **To Next** option to extrude to the closest intersecting face or plane. See **Figure 5-14**. A direction icon indicates where the extrusion will occur. If the default direction is incorrect, pick the alternate direction button. If the extrusion does not terminate at the correct location, you may be able to solve the problem by picking the **Termination** button and picking a different object.

Select the **To** extents option to extrude to a selected face, plane, or point. See **Figure 5-15**. The **Select surface to end the feature creation** button is active, allowing you to pick the face, plane, or point at which to terminate the extrusion. If the location does not intersect the extrusion path, as is common, select the **Terminate feature on extended face** check box.

Figure 5-12.
A—Autoprojecting and constraining all feature edges during sketch creation.
B—Using the automatic sketch profile to create a tapered extrusion.

Automatically projected edges shown in color

A   B
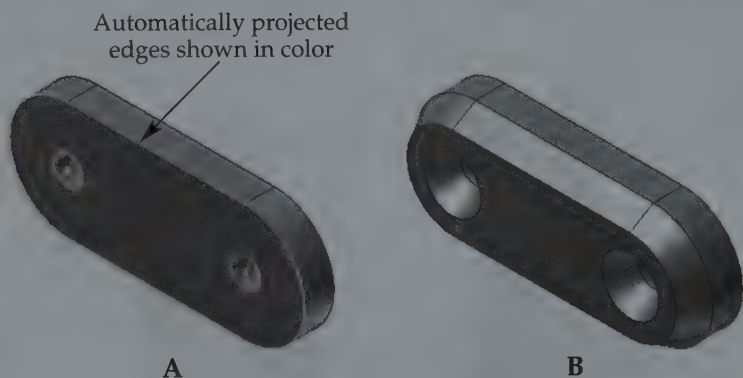
Figure 5-13.
Pick the appropriate operation button to join, cut, or intersect a secondary extrusion.

Vertical
points
fx:.125
fx:.125
Horizontal
points
.125
.500
fx:.125

Sketch Profile

Join

Cut

Intersect
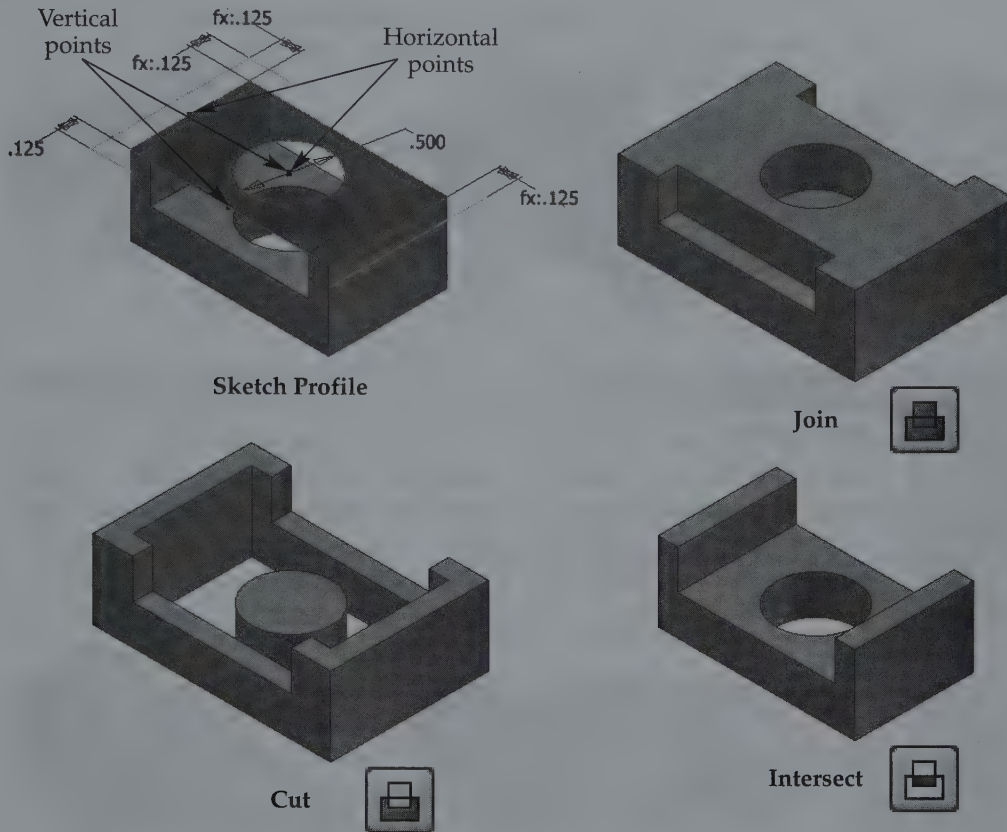
Figure 5-14.
Creating an extrusion using the **To Next** extents option.

Extrusion ends
at the first acceptable
face or plane

Sketch
profile

.950

Extents
To Next
Terminator

Match shape
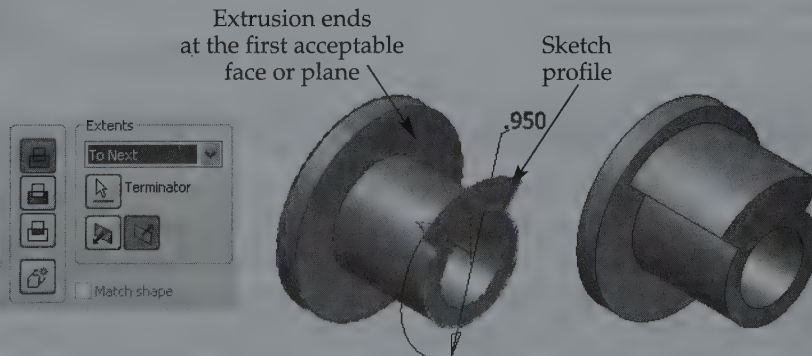
Pick the **From To** extents option to choose a face or plane to begin and end the extrusion. See **Figure 5-16**. The **Select surface to start the feature creation** button is active, allowing you to pick the face or plane to begin the extrusion. The **Select surface to end the feature creation** button then activates, allowing you to pick the face or plane to end the extrusion. If the surfaces you choose to start and end the extrusion do not intersect the extrusion path, as shown in **Figure 5-16**, check the **Terminate feature on extended face** check boxes.

Figure 5-15.
Creating an extrusion using the **To** extents option.

Extrusion ends at selected face or plane

Sketch profile

.088
.188
.300

Select surface to end the feature creation button

Terminate feature on extended face check box
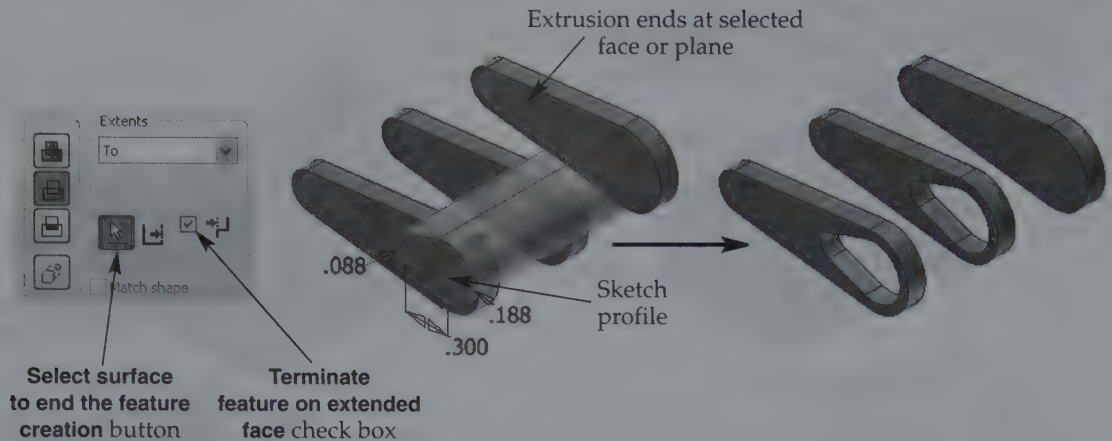
Figure 5-16.
Creating an extrusion using the **From To** extents option.

Select surface to start the feature creation button

Extrusion ends at selected face or plane

Sketch profile

.750

Extrusion begins at selected face or plane

Select surface to end the feature creation button

Terminate feature on extended face check boxes
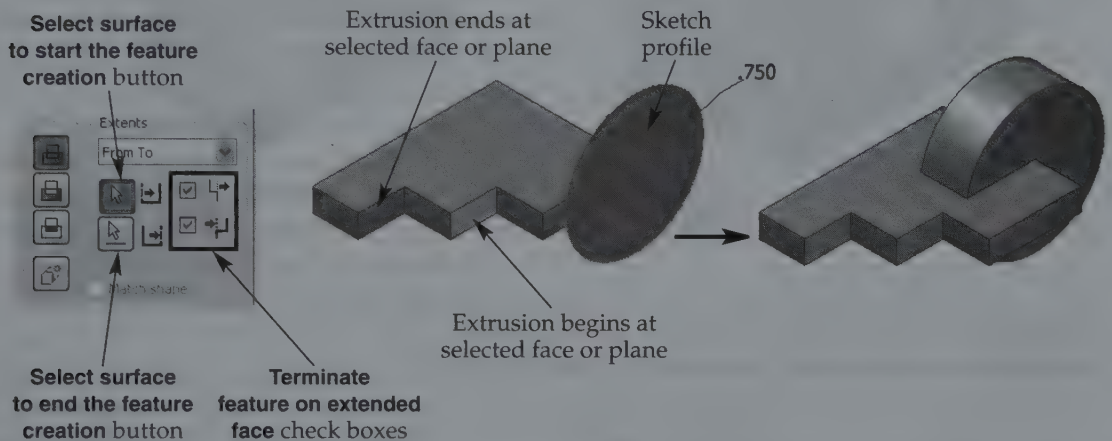
> **NOTE**
>
> If you have difficulty using the **To** or **From To** extents options, you may need to specify additional extents information in the **Alternate Solution** area of the **More** tab. Use the **Flip Direction** buttons to change the direction of the extrusion. You can also select the **Minimum Solution** check box to terminate the extrusion at the nearest plane or face instead of possibly cutting through multiple planes or faces.

Select the **All** extents option to extrude a feature directly through all other features. A distance value and selections are not required. A direction icon indicates where the extrusion will occur. If the default direction is incorrect, pick an alternate direction button. A common application for the **All** extents option is when cutting though the entire model, such as when adding a thru slot. **Figure 5-17** shows an example of using the **All** option to join-extrude a sketch profile to the last possible face.

Figure 5-17.
Creating an extrusion using the **All** extents option.

Extrusion ends at the last
acceptable face or plane

Sketch profile
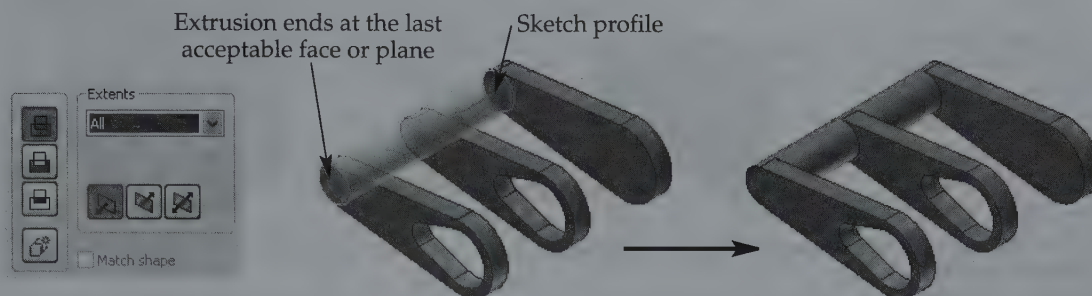
Use the appropriate extents option to ensure parametric relation-
ships. When you use the **To Next**, **To**, **From To**, and **All** extents
options, the length of the extrusion is associated with the begin-
ning and termination faces or planes. This allows the extrusion to
adjust according to model changes, without you having to specify
an existing parameter as the distance.

NOTE

Each feature tool provides unique options. However, the funda-
mental output and operation functions available for the **Extrude** tool
are generally available for other tools. The **Infer iMates** check box is
available when adding many different features to a model. iMates
provide one method of assembling components in an assembly file.

Exercise 5-5

Access the Student Web site (www.g-wlearning.com/CAD) and
complete Exercise 5-5.

Adding Revolutions

The **Revolve** dialog box provides additional, and more usable, options when you
add a revolution to a model with existing features. You can now choose the **Join**, **Cut**,
Intersect, or operation button to add, remove, or combine material using the revolution.
Disregard the **New solid** button for now. The **Full** and **Angle** extents options remain, the
To and **From To** extents options are more usable, and the **To Next** option is available.

Operation and extents options function essentially the same when adding a revo-
lution as when adding an extrusion. Keep in mind however, that revolutions occur
around an axis. **Figure 5-18** shows a few of the many possible applications for adding
revolved features. Study each example to help recognize when it may be appropriate
to use a specific operation and extents combination.

Figure 5-18.
A—A hex screw head created from a base extrusion, followed by a **Cut/Full** revolution. B—A part formed from a base revolution, followed by a **Join/ To Next** revolution. C— A part formed from a base revolution, followed by a **Cut/ Angle** revolution.

Revolution sketch profile (graphics sliced for clarity)

Revolution axis coincident to center point

.059

Extrusion sketch centered on and constrained to projected center point

30.00

Base Distance Extrusion

Cut/Full Extrusion

A

Secondary revolution sketch profile

Secondary revolution uses the same axis as the base revolution

Base Angle Revolution

Join to/Next Revolution

B

.650

.200

Secondary revolution sketch profile (graphics sliced for clarity)

Secondary revolution uses the same axis as the base revolution

90° midplane

Base Full Revolution

Cut/Angle Revolution

C

Exercise 5-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-6.

Supplemental Material

Open Profile Solids

For information about creating extrusions and revolutions using an *open sketch profile*, go to the Student Web site (www.g-wlearning. com/CAD), select this chapter, and select **Open Profile Solids.**

open sketch profile: A sketch profile that does not form a closed loop.

CAUTION

As you learn to use Inventor, you may want to explore tools and techniques for creating multiple solid bodies, and then deriving the bodies to create an assembly and individual part files. However, throughout the exercises and problems in this textbook, ensure that you only create a single solid body per part file.

Coils

A *coil* feature allows you to model items such as springs or replicate thread or helical milling. A coil sketch profile defines the cross section of an individual coil or helical cut. The profile then forms or cuts around an axis according to specified helical parameters. See **Figure 5-19**.

Access the **Coil** tool to create a coil feature using the **Coil** dialog box. See **Figure 5-20**. Fewer options are available when creating a base feature than when building additional features. If a sketch includes one profile, the profile is selected automatically and the **Axis** button is active, allowing you to pick the axis. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile or profiles to coil. You must then select the **Axis** button to pick the axis.
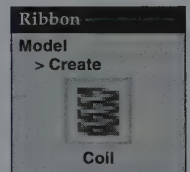
The axis defines the centerline of helix. You must consider the location of the axis when developing a sketch. An axis can be any line in the sketch, an axis in the **Origin** folder of the browser, or a work axis. The sketch in **Figure 5-20**, for example, uses the Y axis, which is appropriate for the sketch profile on the XY plane. If the direction of the coil preview appears incorrect, pick the **Flip** button to change the direction.

coil: A helix or spiral feature used to create springs, detailed threads, and similar items.

Ribbon

Model
> Create

Coil

PROFESSIONAL TIP

Constrain a coil profile to the projected center point when possible to constrain to the origin, not just a point in space. The projected center point is also coincident to the axes in the **Origin** folder of the browser, which you can select as axes for coils.

You can coil a closed profile as a solid or surface by picking the appropriate button in the **Output** area. You can coil an open profile only as a volumeless surface. When creating a solid base feature, the only possible operation is to form a new solid body, as evident by the selected **New Solid** button. When you add a coil, the **Join, Cut**, and **Intersect** operation buttons are available for adding, removing, or combining material using the coil.

Figure 5-19.
A—An example of a sketch profile used to form a base feature coil. This example uses the vertical axis coincident to the projected center point as the axis. B—An example of a sketch profile used to cut threads into an extruded cylinder. This example uses the center line of the cylinder as the axis.

Horizontal constraint

Projected center point

10

30

A

Threads cut through existing cylinder

60,00

.010 .029

60,00

.010 .029

B
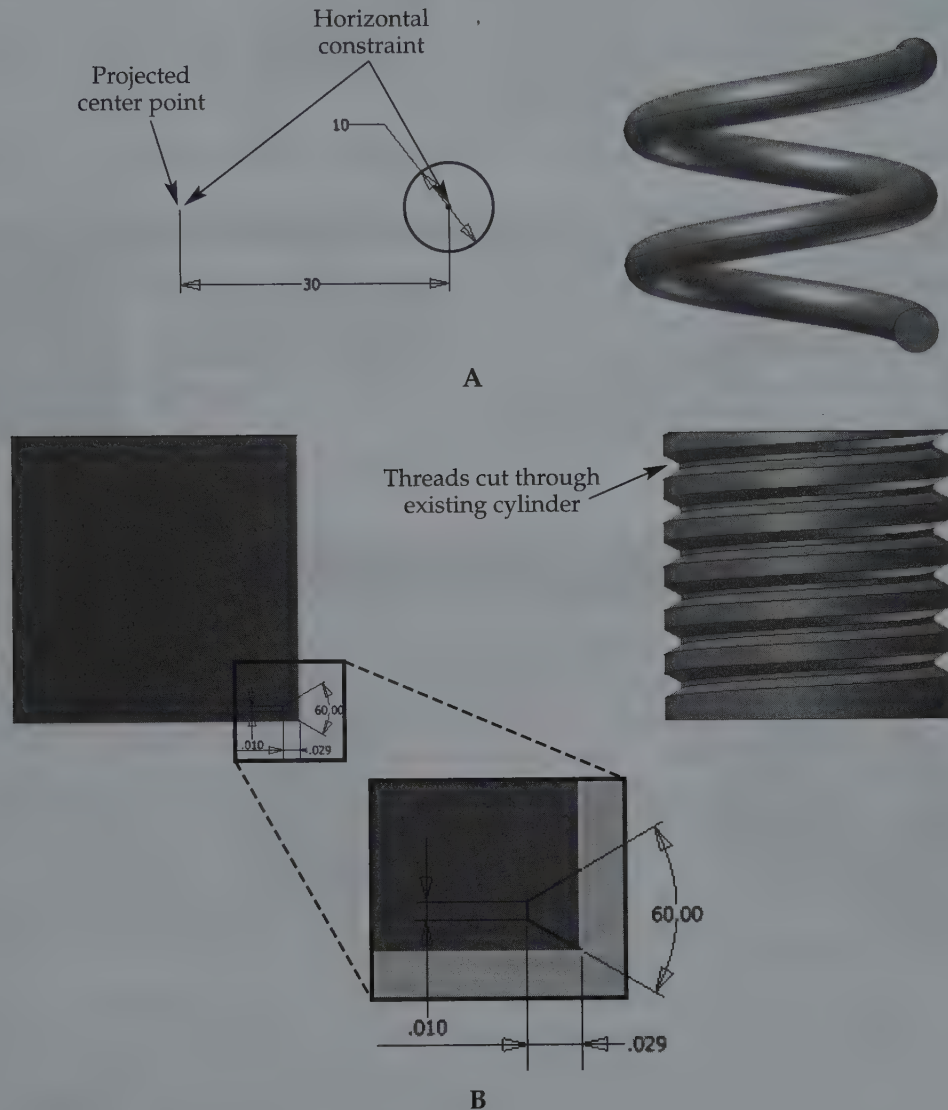
Figure 5-20.
The **Coil** dialog box allows you to select a sketch profile and specify coil parameters and settings. The **Coil Shape** tab controls primary coil shape characteristics. The preview provides a good idea of how the coil will form and helps you select appropriate options.

Coil : Coil

Coil Shape Coil Size Coil Ends

Flip button

Profile

Axis

Solid

Output

Rotation

Selected axis coincident to center point

Selected sketch profile constrained to center point

25   10

OK   Cancel
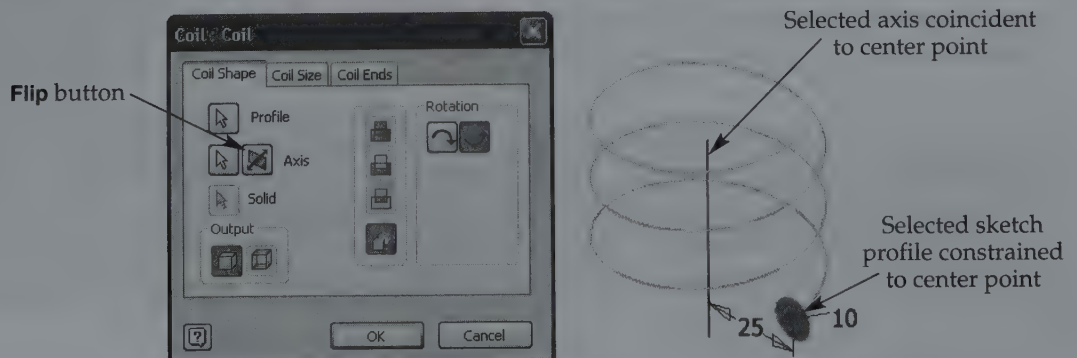
Specify the coil rotation by picking the **Clockwise** or **Counterclockwise** button in the **Rotation** area. A counterclockwise rotation corresponds to right-hand threads. A clockwise rotation corresponds to left-hand threads.

Coil Size

Pick the **Coil Size** tab to specify the size of the coil. See **Figure 5-21**. Pick an option from the **Type** drop-down list to create a coil according to known parameters. Select the **Pitch and Revolution** option to specify the *pitch* and number of coil *revolutions*. Choose the **Revolution and Height** option to identify the number of revolutions and *height*. Select the **Pitch and Height** option to identify the pitch and height. Use the text boxes that appear when you choose a type option to specify values.

The **Taper** text box is available when you select a coil type other than **Spiral**. Use the **Taper** text box to specify a taper angle for the coil. See **Figure 5-22**. Select the **Spiral** type option to create a coil without height. See **Figure 5-23**. Use the **Pitch** and **Revolution** text boxes to specify the spiral parameters.

pitch: The distance, parallel to the axis, between a point on one coil revolution to the corresponding point on the next coil revolution.

revolution: In a coil, one complete spiral, or 360° loop.

height: In a coil, the total length of the coil from a point at the beginning of the coil to a corresponding point at the end of the coil.

Figure 5-21.
The **Coil Size** tab of the **Coil** dialog box provides options for specifying the coil pitch, height, taper, and number of revolutions, depending on the coil type.
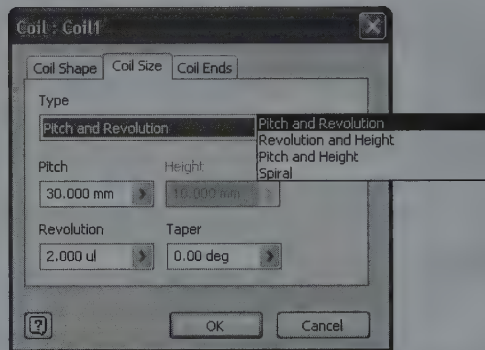
Figure 5-22.
An example of a coil with a 45° taper.
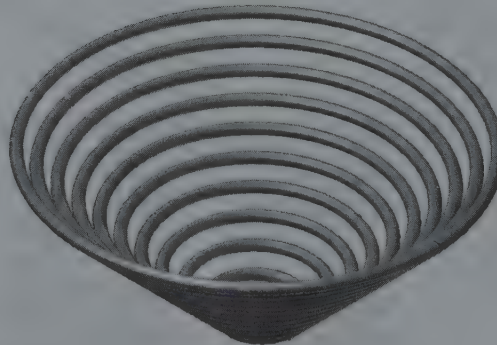
Figure 5-23.
An example of a spiral coil.
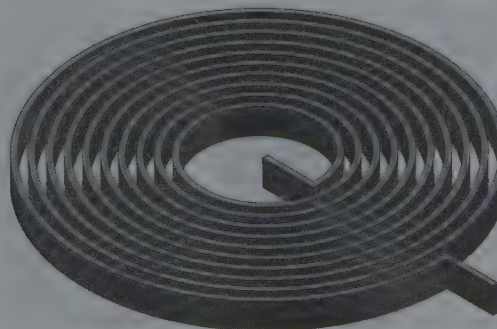
Coil Ends

Pick the **Coil Ends** tab to specify the formation of coil ends. See **Figure 5-24**. You can adjust the appearance of the start and end of the coil independently using options in the **Start** and **End** areas. Use the default **Natural** option to use a *natural end,* or pick the **Flat** option to apply a *flat end.*

A natural end is unaltered, or untreated, and uses 0° transition angles. Selecting the **Flat** option activates the **Transition Angle** and **Flat Angle** text boxes used to specify the *transition angle* and *flat angle.* Coil1 in **Figure 5-24** shows the difference between a natural end and an end with a 90° transition angle and 90° flat angle. Coil2 in **Figure 5-24** shows a transition angle of 0° and a flat end of 180°. Pick the **OK** button to create the coil feature.

NOTE

Applying transition and flat angles to a coil does not change the profile geometry.

PROFESSIONAL TIP

Use the **Coil** tool to create realistic, detailed threads when necessary. However, for most applications, the **Thread** tool is adequate for generating thread representations.

Exercise 5-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 5-7.

Figure 5-24.
The **Coil Ends** tab of the **Coil** dialog box includes options for specifying start and end transitions.
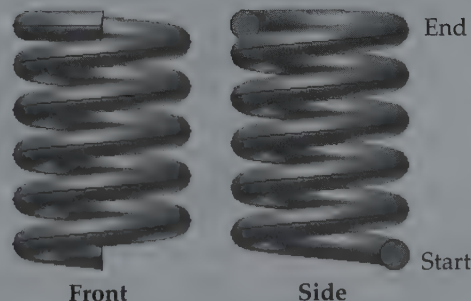
Coil1
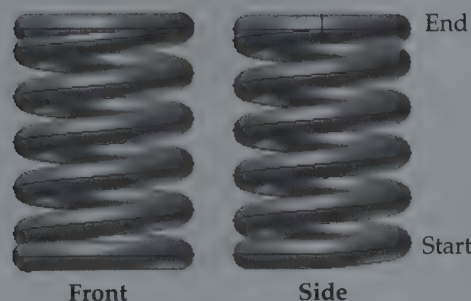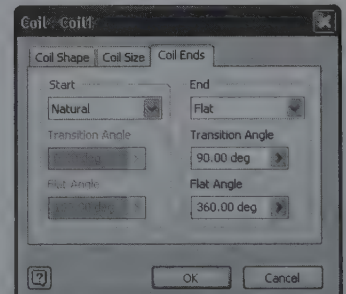
Coil2
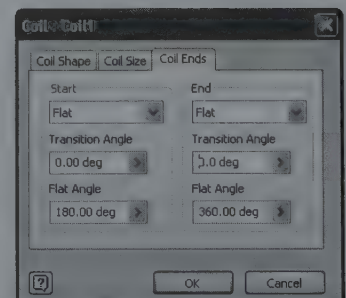
Editing Features

To edit a feature, double-click on the feature in the browser or right-click on the feature in the browser and pick **Edit Feature**. The associated feature dialog box reappears, allowing you to make changes. For example, the **Extrude** dialog box displays when you edit an extrusion. See **Figure 5-25A**. The consumed sketch and other associated content, such as work features, also control the size, shape, and location of a sketched feature. Edit the sketch to make changes to the feature that you cannot make using the feature dialog box. See **Figure 5-25B**.

You can adjust the design history by dragging and dropping sketches and features above or below certain items in the browser. Drag **End of Part** above an existing feature, or drag features below **End of Part**, to observe the design history or add a feature at an earlier point in the design. For example, **Figure 5-26** shows adding a collar to a T pin early in the design. Save the original part as a new design to create a new part while keeping the original alternative design. In the redesign file, drag and drop **End of Part** above the extrusion used to create the pin, and then create the collar. Notice how the original feature names remain. To complete the collared T pin, you must redefine and constrain the sketch plane used to position the pin, because the collar feature removes the correct association.

Figure 5-25.
A—An example of editing a feature to adjust a model parameter: the length of the T pin shaft.
B—An example of editing a sketch to adjust a model parameter: the diameter of the T pin handle.
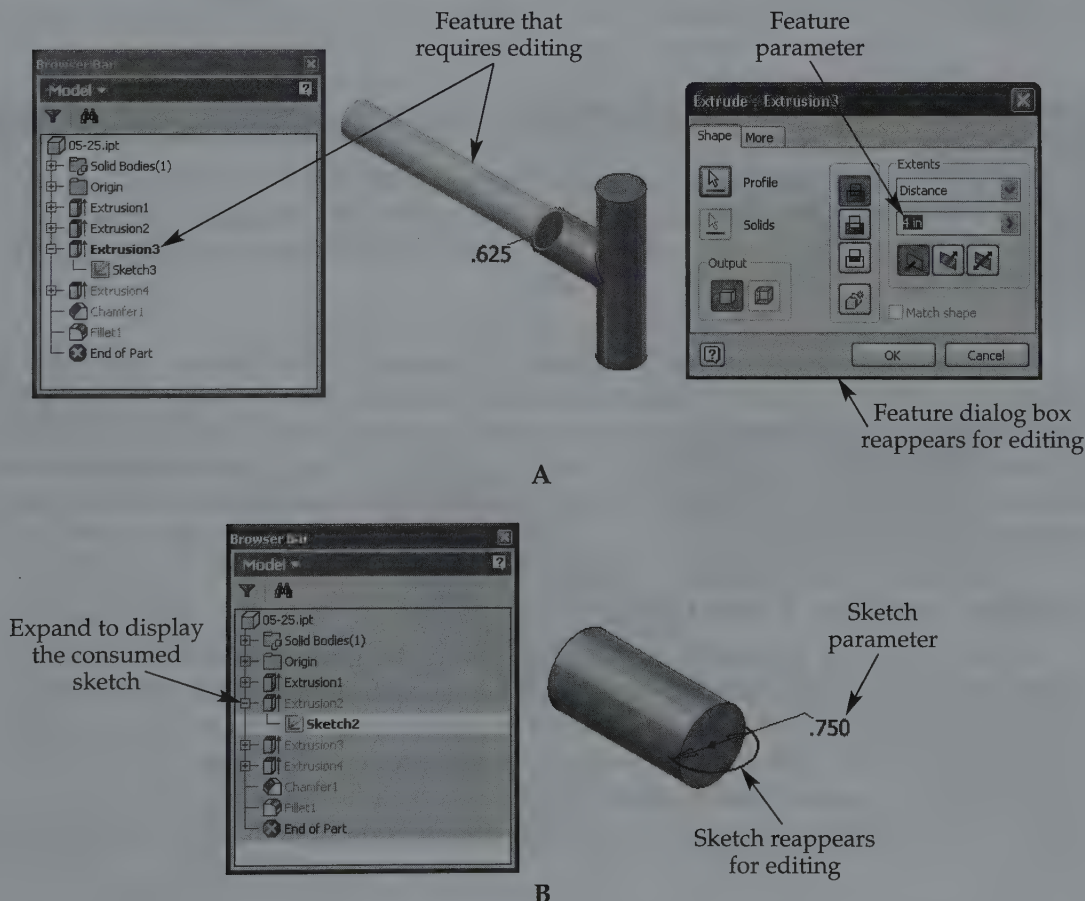
Figure 5-26.
An example of modifying and adding features to create a new or different design. Moving the **End of Part** indicator in the browser temporarily turns off items in the model.

Pin

Original Part

Pin sketch redefined
and reconstrained to
new (collar) plane

Feature edited to add
fillets and rounds

Collar

End of Part dragged above
where new features were added

End of Part
returned to the end

New Part
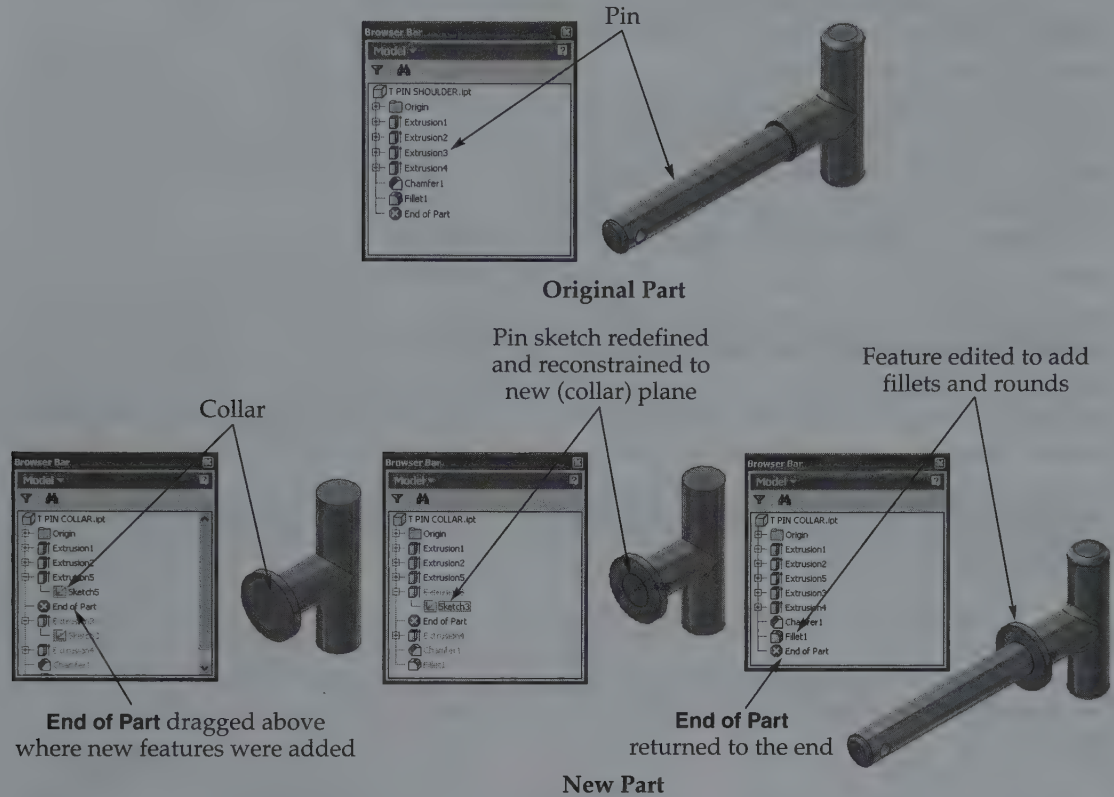
CAUTION

You cannot rearrange features in the browser that must occur before or after other features. Adding new features before existing features often results in errors if the new features remove or alter existing feature reference geometry.

Delete an unneeded feature by picking the feature in the browser and pressing [Delete] or right-clicking on the feature in the browser and selecting **Delete**. A dialog box appears, allowing you to keep items consumed by or that reference the feature. Deselect the check boxes corresponding to items you want to keep. Most model content is linked together. When you remove a sketch or feature, the model often loses associated content.

Feature and Face Properties

Right-click on a feature in the browser or in the graphics window and select **Properties** to display the **Feature Properties** dialog box. See **Figure 5-27**. The options in the **Feature Properties** dialog box are specific to the selected feature and vary depending on the feature type. Most features include a **Name** property that you can change to a more descriptive name. You can also change the default feature name, as well as most other items in the browser, by slowly double-clicking on the item and entering a new name.

Figure 5-27.
The **Feature Properties** dialog box controls feature-specific properties. This example shows the properties of an extrusion renamed **BASE**.

Selected feature   Change the feature name

Feature suppression options

Pick a unique color style
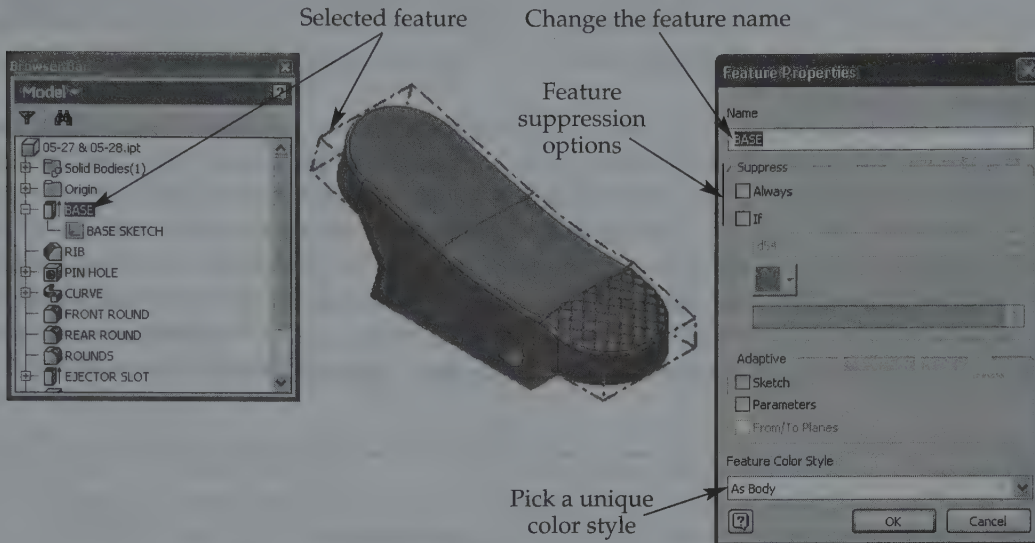
The **Suppress** area contains options for suppressing, or turning off, the selected feature. Feature suppression is commonly used to explore design ideas or to allow the model to adapt to other features or assembly components. The model does not calculate a suppressed feature. You can also suppress a feature by right-clicking on the feature in the browser or in the graphics window and picking **Suppress Feature**. This is the same as selecting the **Always** check box in the **Feature Properties** dialog box.

To change the color of the part, select a color from the **Feature Color Style** drop-down list. The color of a feature is often set to a different color to represent a special surface treatment or finish, or to emphasize a feature for modeling purposes. To change the color of a specific face, close the **Feature Properties** dialog box. Then right-click on a face and select **Properties** to display the **Face Properties** dialog box. Pick the face color from the **Face Color Style** drop-down list. See **Figure 5-28**.

Figure 5-28.
Use the **Face Properties** dialog box to change the color style applied to selected model faces. This example shows a knurled color style override applied to model faces to represent a knurled surface finish.

Pick a unique color style

Selected face
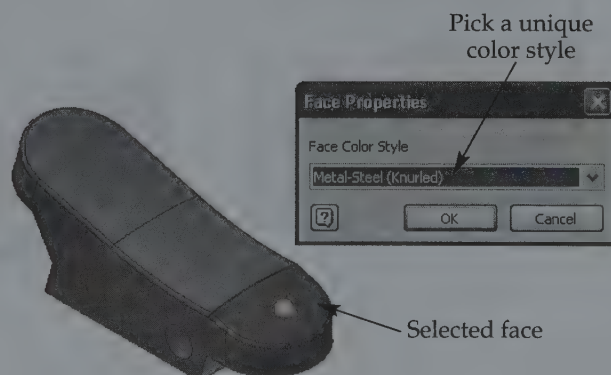
Feature Parameters

A model receives parameters when you add dimensional constraints to a sketch and when you create a feature. For example, an extrusion includes parameters associated with sketch profile dimensional constraints, as well as extrusion extents and the taper specified in the **Extrude** dialog box. Use the **Parameters** dialog box to help manage model parameters.

Figure 5-29 shows an extrusion and the corresponding **Parameters** dialog box. The figure includes the **Extrude** dialog box to show the application of the THICKNESS parameter. In this example, parameter d4 is the extrusion distance and d5 is the taper angle. These are the only two parameters specifically associated with the feature. All other parameters control the size and shape of the sketch. Notice that the user parameter THICKNESS, which is an equation linked to the parameter d4, controls the extrusion distance. Select the THICKNESS parameter by picking the **List Parameters** option from the **Distance** text box in the **Extrude** dialog box instead of entering a value.

Figure 5-29.
Working with model parameters in the **Parameters** dialog box. The **Extrude** dialog box is shown for reference only. You must complete an extrusion to view parameters in the **Parameters** dialog box.
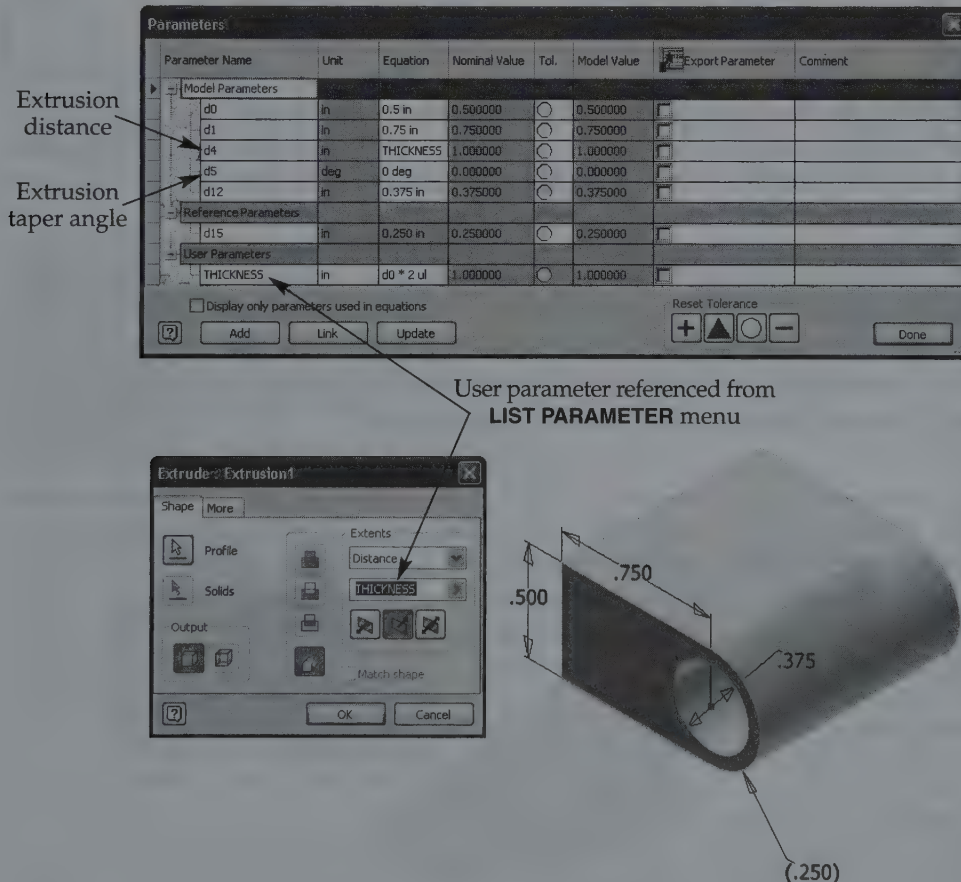
Chapter Test

1. What is a base feature?
2. Define *extrusion*.
3. What is a surface extrusion?
4. Name the default orbit mode for viewing an object.
5. Briefly describe how to reuse a consumed sketch.
6. Define *revolution*.
7. What is the difference between using the **Full** and **Angle** extents options to create a revolution?
8. Briefly explain the purpose of extruding an object using a cut operation.
9. What is a coil?
10. What defines the centerline of a helix?
11. Define the terms *pitch* and *revolution* as they relate to coils.
12. Explain how to edit a feature.
13. How do you access the **Feature Properties** dialog box?
14. Explain the purpose of suppressing a feature, and give an application for feature suppression.
15. Name the dialog box in which model parameters are controlled.

Problems

1–8 Instructions:

- *Open the specified Chapter 4 part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

▼ Basic

1. **File:** P4-1.ipt
   Save as: P5-1.ipt
   Title: PIN
   Specific instructions: Extrude the sketch profile 52 mm using the **Midplane** option, as shown.

▼ Basic

2. **File:** P4-2.ipt
   Save as: P5-2.ipt
   Title: SWIVEL
   Specific instructions: Extrude the sketch profile 9.5 mm in the positive direction as shown.
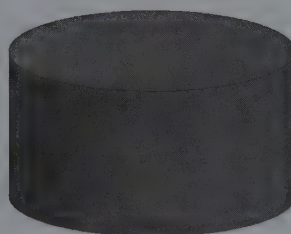
3. **File:** P4-3.ipt
   Save as: P5-3.ipt
   Title: SCREW
   Specific instructions: Extrude the sketch profile 80 mm in the positive direction as shown. Open a sketch on the bottom face of the extruded cylinder and sketch a ∅5.35 circle coincident to the center point of the bottom face. Change the projected edge of the cylinder to a construction linetype so that Inventor recognizes only the larger circle as the sketch profile. Extrude the sketch 7.2 mm away from Extrusion1 as shown. Open a sketch on the top face of the extruded cylinder and sketch a ∅12.7 circle coincident to the center point of the top face. Change the projected edge of the cylinder to a construction linetype so that Inventor recognizes only the smaller circle as the sketch profile. Extrude the sketch 22 mm away from Extrusion1 as shown.
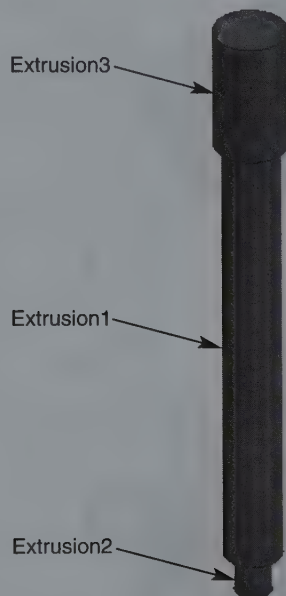
Extrusion3

Extrusion1

Extrusion2

4. **File:** P4-4.ipt
   Save as: P5-4.ipt
   Title: SUPPORT BRACKET
   Specific instructions: Extrude Profile1 .125″ midplane. Share Sketch1. Extrude the three circle profiles (Profile2) .375″ midplane. Open a sketch on **Face1** and offset the automatically projected edges .0625″. You may need to edit and constrain sketch geometry. Cut-extrude the sketch through the entire part as shown.
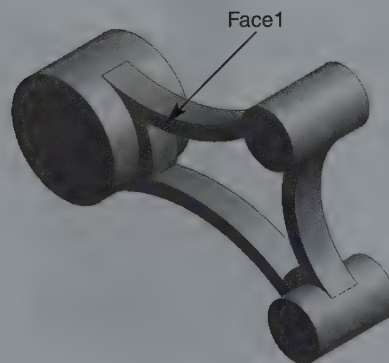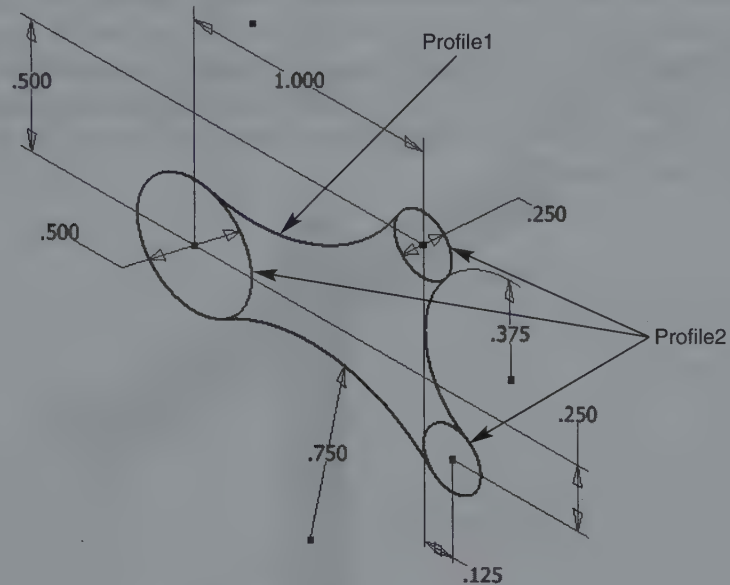
5. **File:** P4-6.ipt
   Save as: P5-5.ipt
   Title: BODY
   Specific instructions: Extrude the sketch 19 mm midplane as shown.

6. **File:** P4-7.ipt
   Save as: P5-6.ipt
   Title: HEAD
   Specific instructions: Revolve the sketch 360° around the Y axis as shown.

7. **File:** P4-8.ipt
   Save as: P5-7.ipt
   Title: HANDLE
   Specific instructions: Extrude the outside sketch profile .65" midplane as shown.
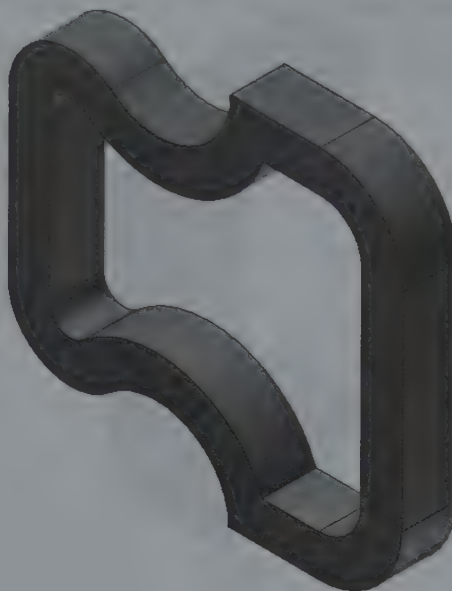
8. **File:** P4-9.ipt
   Save as: P5-8.ipt
   Title: STOP
   Specific instructions: Extrude the sketch .288" midplane as shown.
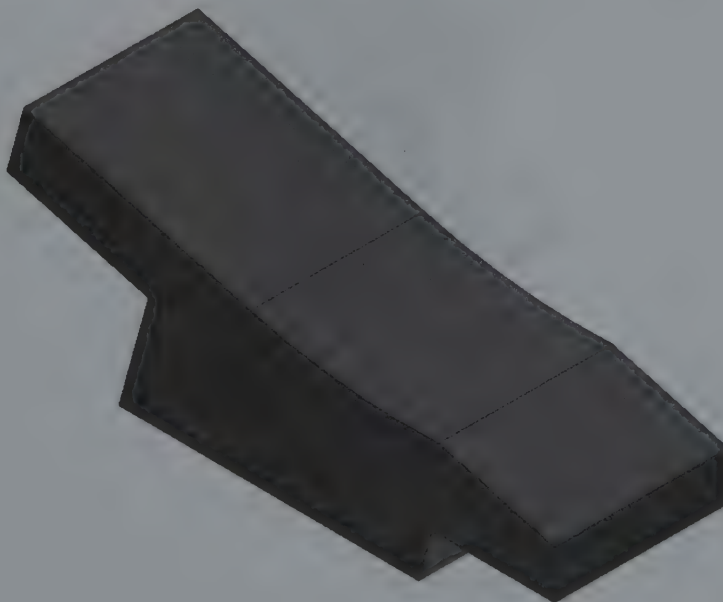
9–12 Instructions:

- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects at the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Do not assign material and color properties at this time.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

9. **Title:** FUNNEL
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-008-01
   Project: FUNNEL
   Sketch plane: XY
   Save as: P5-9.ipt
   Specific instructions: Create the sketch profile shown on the XY plane. Fully revolve the sketch around the centerline sketch line. Sketch a rectangle tangent to the radius of the circle on the top face of the revolution. Change the projected edge of the cone to a construction linetype so that Inventor recognizes only the square as the sketch profile. Extrude the sketch .125″ toward the revolution as shown. Be sure the extrusion adds material through the revolution.
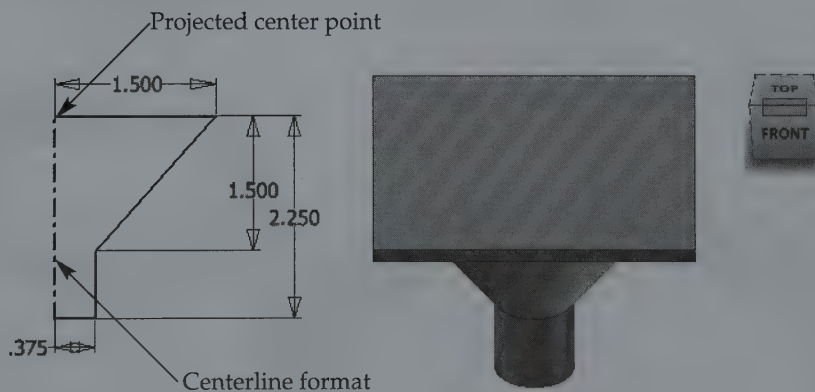
10. **Title:** SWIVEL BOLT
    Units: Inch
    Template: Part-IN.ipt
    Part Number: IAA-009-02
    Project: TORSION SPRING
    Sketch plane: XY
    Save as: P5-10.ipt
    Specific instructions: Create the sketch profile shown on the XY plane. Fully revolve the sketch around the Y axis as shown.

.125

1.000

60.00

Projected center point

.250
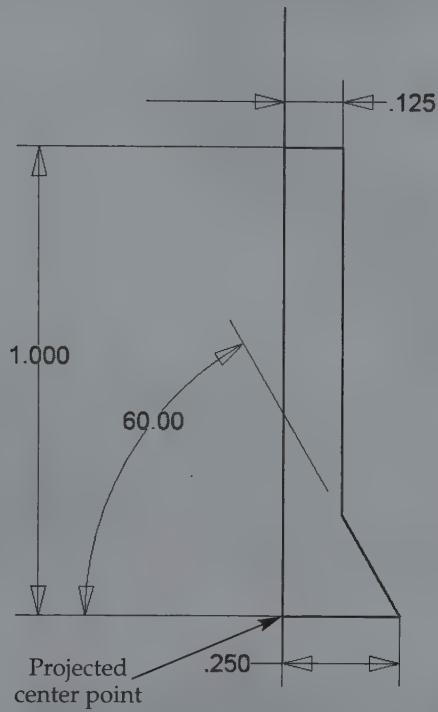
11. **Title:** SWIVEL EYE
    Units: Inch
    Template: Part-IN.ipt
    Part Number: IAA-009-03
    Project: TORSION SPRING
    Sketch plane: XY
    Save as: P5-11.ipt
    Specific instructions: Create the sketch profile shown on the XY plane. Revolve the sketch 305° around the Y axis. Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of Face1. Extrude the sketch .75″ away from the revolution. Use the **Autoproject edges for sketch creation and edit** tool to create a sketch of Face2. Add a vertical center line .5″ from the center of the sketched circle as shown. Revolve the sketch 320° around the sketched centerline. The final part should look like the part shown.

Horizontal
.125
Projected center point
.250
Face1, original sketch plane (behind extrusion)
Axis line
Face2
.500
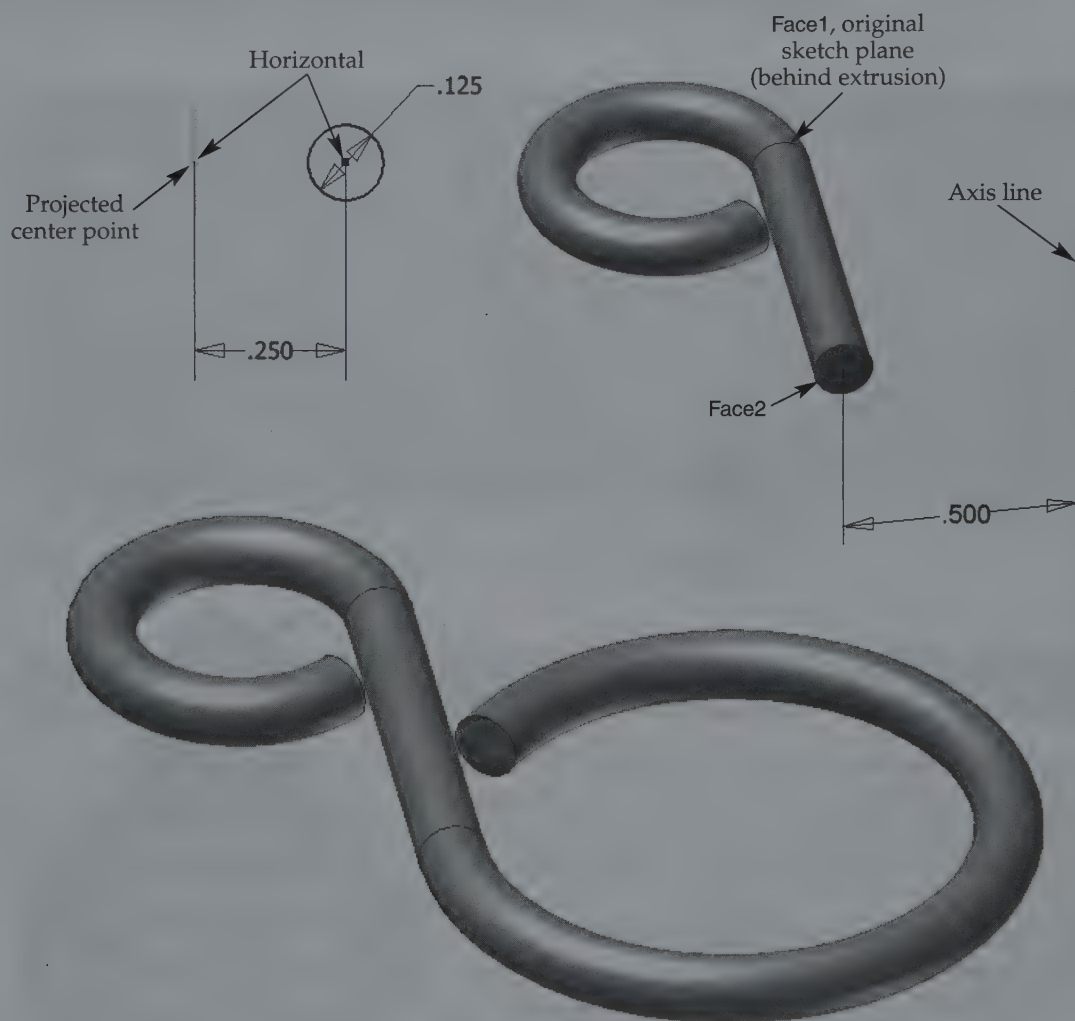
12. **Title:** TORSION SPRING
    Units: Inch
    Template: Part-IN.ipt
    Part Number: IAA-009-01
    Project: TORSION SPRING
    Sketch plane: XY
    Save as: P5-12.ipt

Problems - Chapter 5

Specific instructions: Create each of the coils as described below. Use the Autoproject edges for sketch creation and edit tool to create the sketches for Coil2 through Coil6. Use the Y axis and a right-hand coil for all six coils, and refer to the parameters shown in the table. The final problem should look like the part shown.

Coil1: Create the sketch profile shown on the next page on the XY plane. Coil the sketch in a positive direction.

Coil2: Create a sketch of the bottom coil face. Coil the sketch in a negative direction.

Coil3: Create a sketch of the top coil face of Coil1. Coil the sketch in a positive direction.

Coil4: Create a sketch of the top coil face of Coil3. Coil the sketch in a positive direction.

Coil5: Create a sketch of the top coil face of Coil4. Coil the sketch in a positive direction.

Coil6: Create a sketch of the top coil face of Coil5. Coil the sketch in a positive direction.

	Coil1	Coil2	Coil3	Coil4	Coil5	Coil6
Pitch	.125″	.125″	.25″	.25″	.125″	.125″
Revolutions	5	5	5	5	5	3
Taper	0°	–30°	30°	–30°	0°	–30°
Start	Natural	Natural	Natural	Natural	Natural	Natural
End	Natural	Flat, 90° transition angle, 0° flat angle	Natural	Flat, 90° transition angle, 0° flat angle	Natural	Flat, 0° transition angle, 45° flat angle
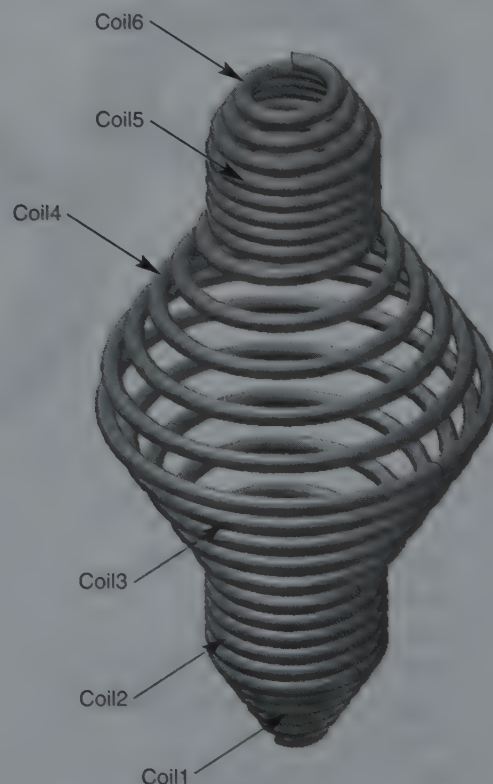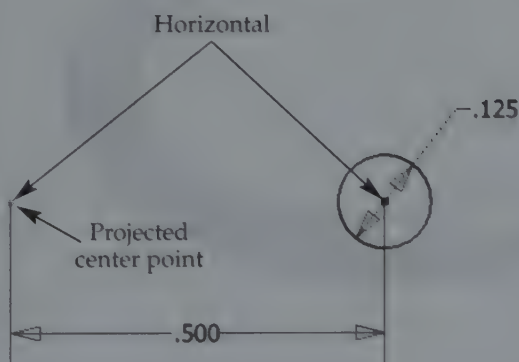
View Tools and Design Properties

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Use display and navigation tools to position models on-screen.
✓ Adjust display options and characteristics.
✓ Choose color, material, and lighting styles.
✓ Use basic model inspection tools.

View, display, and navigation tools are critical to the entire design and documentation process. This chapter focuses on the tools and options needed to orient and view a model in 3D space. You will use many of the same view tools to create sketches and prepare 2D drawings. This chapter also explains model color, material, and lighting styles, and describes basic tools for inspecting model geometry.

View Tools

View tools allow you to navigate the 3D model environment and observe detailed objects in 2D environments. Understanding how to use view tools to control the appearance of items on-screen is critical, especially after you create model features. Most common modeling tasks require that you rotate, zoom, and pan to orient a model and display design characteristics.

View tools do not change the physical location, size, or shape of objects. Some view tools adjust the display according to *screen space*, while other tools recognize *model space*. The **View** ribbon tab supplies most view tools. However, the **Navigation Bar**, shortcut menus, and keyboard shortcuts often provide a much faster method of accessing common view tools.

screen space:
A space, or environment, in which the graphics window controls model display; the center is the center of the graphics window.

model space:
A space, or environment, in which the model defines the display orientation, regardless of the position of the model in the graphics window; the model pivot point defines the center.

Navigation Bar

The **Navigation Bar** is a view toolbar positioned near the upper-right corner of the graphics window by default. See **Figure 6-1**. The **Navigation Bar** is available in all file types and includes tool buttons for accessing common view tools associated with the current file type.

Ribbon

View
> Windows

User Interface
> Navigation Bar
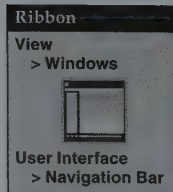
Figure 6-1.
Use the **Navigation Bar** to access common view tools. Pick a tool button or select an option from a flyout to initiate the corresponding tool.

Pick to close

Flyout

Pick to access the menu of basic adjustments shown

Check a tool to add to the navigation bar

✓ ViewCube (active)	
✓ SteeringWheels	
✓ Pan	
✓ Zoom	
✓ Orbit	
✓ View Face	Page Up
Zoom All	Home
Zoom Window	Z
Zoom Selected	End
Previous View	F5
Next View	Shift+F5
Projection	
Shading	
Docking positions	▶
Inactive opacity	▶
Active opacity	▶

Navagation bar display options
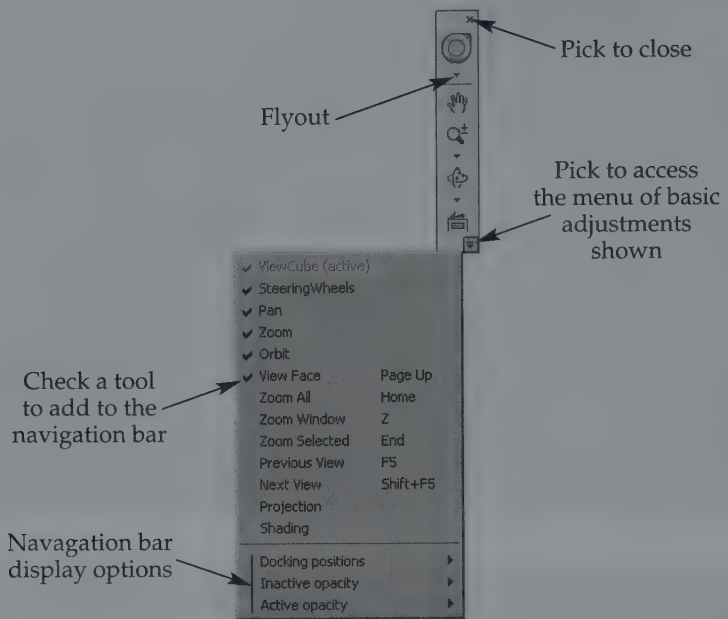
To add tools to the **Navigation Bar**, pick the **Customize** flyout shown in **Figure 6-1** and select a tool from the menu. The flyout also includes basic adjustment options. Select a docking position from the lower portion of the **Docking positions** cascading menu to specify the location of the **Navigation Bar** in the graphics window. The checked **Linked to ViewCube** option means that the ViewCube, shown above the **Navigation Bar** by default, always appears near the **Navigation Bar**. The lower the selected **Active opacity** or **Inactive opacity** percentage, the more transparent the **Navigation Bar** appears when it is active and when it is not active (when the cursor is not hovering over the bar).

A tool or option accessible from the **Navigation Bar** appears as a graphic in the margin of this textbook. The graphic represents the process of picking a **Navigation Bar** button or an option from a flyout. The example shown in this margin illustrates accessing the **Pan** tool from the **Navigation Bar**.

Zooming and Panning

Several zoom tools allow you to increase or decrease the displayed size of objects in the graphics window. Access the **ZOOM** tool to change the display size using *realtime zooming*. Press and hold the left mouse button and move the cursor away from the model to zoom in or toward the model to zoom out. When you achieve the correct display, release the mouse button. Press [Esc], pick in the graphics window, right-click and pick **Done**, or access another tool to exit.

Navigation Bar

Pan

Navigation Bar

Zoom

Ribbon

View
> Navigate

Zoom

realtime zooming: Zooming that you can view as it is performed.

NOTE

You can reverse the realtime zoom direction by picking the **Reverse Direction** check box in the **3D Navigation** area on the **Display** tab in the **Application Options** dialog box.

PROFESSIONAL TIP

By default, roll a mouse wheel forward (away from you) to zoom out, and roll the wheel backward (toward you) to zoom in. This function also pans to the location of the cursor during zoom operations.

Access the **Zoom Window** tool to create a window around the area to zoom. Pick a start point and then pick a diagonal point. The actual size of the window box corresponds to the height and width ratio of the graphics window. See **Figure 6-2**. Use the **Zoom Selected** tool to zoom to a selected curve or face as shown in **Figure 6-3**. If you pick a point, the display pans to position the selected point in the center of the graphics window. Use the **Zoom All** tool to view all objects in the graphics window.

Access the **Pan** tool to pan the display in the graphics window. Click and hold while moving the **Pan** icon. To use the [F2] key to pan, press and hold the key while moving the mouse. You can also pan by pressing and holding the wheel of a wheel mouse or by pressing arrow keys.

Exercise 6-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 6-1.

Figure 6-2.
Pick two points or drag to create a window around the area to zoom.

Pick first

Pick second

Zoomed view fills the graphics window
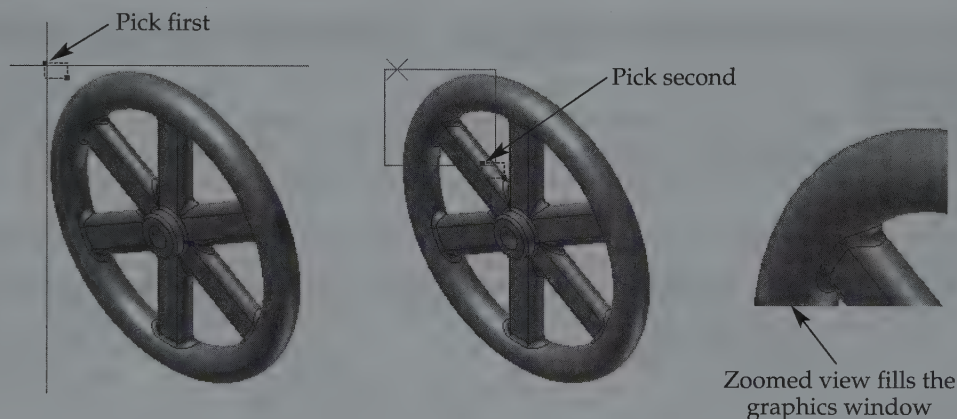
Figure 6-3.
After picking **Zoom Selected**, pick an object to zoom.

Pick object

Zoomed view fills graphics window

Navigation Bar
Zoom Window
Ribbon
View
> Navigate
Zoom Window
Type
[Z]

Navigation Bar
Zoom Selected
Ribbon
View
> Navigate
Zoom Selected
Type
[End]

Navigation Bar
Zoom All
Ribbon
View
> Navigate
Zoom All
Type
[Home]

Navigation Bar
Pan
Ribbon
View
> Navigate
Pan

Orbiting

Free Orbit

Ribbon

View
> Navigate

Free Orbit

Type

[F4]

Navigation Bar

Constrained Orbit

Ribbon

View
> Navigate

Constrained Orbit

pivot point: The point that acts as the center point when you are viewing and rotating model space objects.

Use the **Orbit** tool to view a model from any angle. The **Orbit** tool is set to **Free Orbit** mode by default. A **Constrained Orbit** mode is also available. Both modes function essentially the same and use the same on-screen symbols. See **Figure 6-4**. The difference between the two modes is how Inventor interprets the center of the orbit, or *pivot point*, and how you look at the model. The **Free Orbit** mode considers screen space, while the **Constrained** mode recognizes model space. If you think of a model as an actual product, orbiting in **Free Orbit** mode appears as if the product is orbiting in front of you, while you remain stationary. Orbiting in **Constrained** mode appears as if you are moving around the product to view different angles, while the object remains stationary.

Figure 6-4 shows the behavior of the **Orbit** tool. Hold down the left mouse button and drag the orbit icon to orbit freely. To orbit around the vertical axis line, drag one of the horizontal lines left or right. To orbit around the horizontal axis line, drag one of the vertical lines up or down. To orbit around the horizontal axis point, drag near an outside quadrant. You can also use the **Orbit** tool to pan the display. Instead of holding down the left mouse button to drag, pick a point to pan the display away from the selection. Press [Esc], pick outside the rotation area, right-click and pick **Done**, or access another tool to exit.

> **NOTE**
>
> Define the default orbit mode by picking the **Free** or **Constrained** radio button in the **3D Navigation** area on the **Display** tab of the **Application Options** dialog box.

Figure 6-4.
The position of the cursor determines the method of rotation when you use the **Orbit** tool.
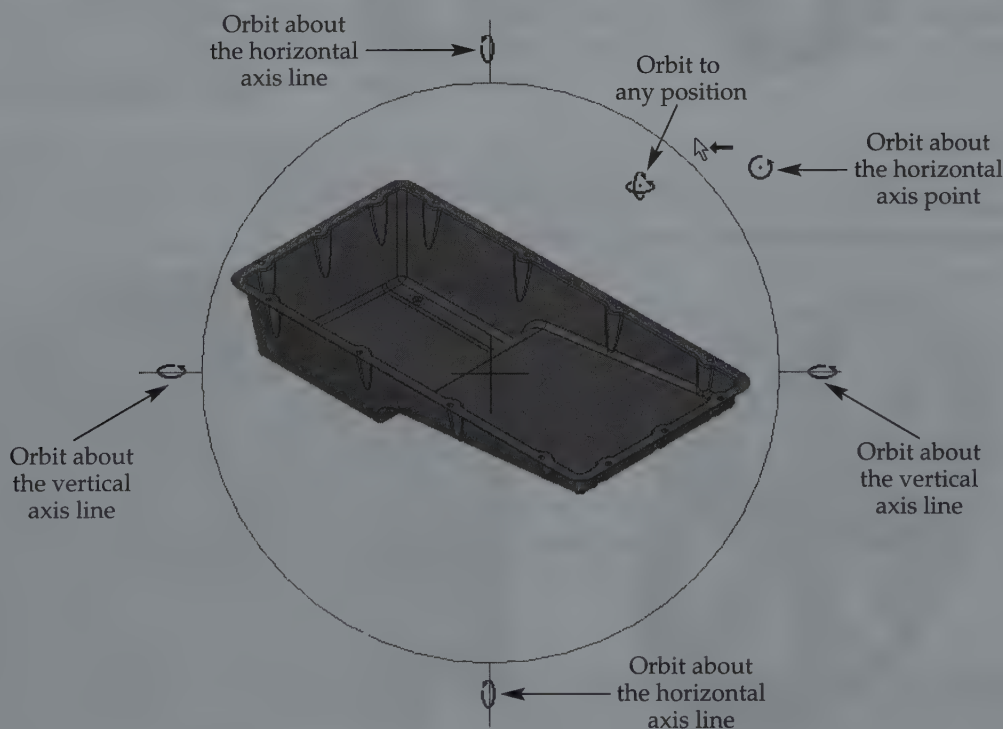
To rotate the model in a continuous display, access the **Orbit** tool, hold down the [Shift] key, and pick a location to begin the orbit. Then move the cursor, release the mouse button, and then release the [Shift] key. The model continues to orbit. The faster you move the cursor and release the mouse button, the faster the continuous orbit. You can use other view tools while continuously orbiting.
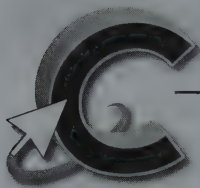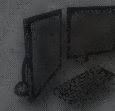
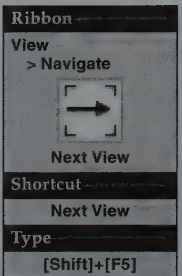## Exercise 6-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 6-2.

ViewCube

The ViewCube appears in the upper-left corner of the graphics window, above the **Navigation Bar**, in all model work environments by default. See **Figure 6-5**. The View-Cube allows you to view a model precisely using a labeled cube. Precise positions include *orthographic*, *isometric*, and a form of *oblique* view angles.

To use the ViewCube, pick a face, edge, corner, or arrow on or around the cube to adjust the display. The cube and the model rotate together. If you pick a model object such as a point, edge, or face *before* selecting a location on the ViewCube, the view zooms in on the object in addition to displaying the selected cube orientation.

orthographic view: A 2D view, or projection, in which the line of sight is perpendicular to a surface, such as the front of an object or the XY plane.

isometric view: A 3D view in which all three axes appear at equal (120°) angles with the plane of projection.

oblique view: A 3D view in which the plane of projection is parallel to the front surface, and a receding angle is applied.

Figure 6-5.
The ViewCube is an excellent tool for adjusting the displayed rotation of a model to a precise view position, including orthographic, isometric, and oblique views. You can also orbit using the ViewCube.

Pick to display the default, or home, view

Pick any corner to display the corresponding isometric view

Pick one of the arrows to display the corresponding orthographic view

Pick to rotate the display 90°

Pick to access options for adjusting the ViewCube

Pick any face to display the corresponding orthographic view

Isometric View

Orthographic View

Pick any edge to display the corresponding oblique view

Drag any corner, edge, or face to orbit

Oblique View

Orbiting

To look at an isometric view, pick a cube corner. The isometric view in **Figure 6-5** is the default XY isometric orientation. Use the **Home View** tool to return the display to the default isometric view. To look at an orthographic view, pick a cube face. The orthographic view in **Figure 6-5** is the FRONT orientation. While displaying an orthographic view, pick the arrows around the cube to view a different orthographic view, or select the clockwise or counterclockwise arrow to rotate the display 90°. You can also pick a corner or edge to rotate to an isometric or oblique view. To display a view best described as an oblique view with a 0° receding angle, pick a cube edge.

PROFESSIONAL TIP

Consider the orientation of the model and the plane on which you sketch the base feature to ensure that the ViewCube faces correspond with the appropriate orthographic views of the model. For example, by default, the FRONT face matches the XY plane. If you sketch on the XY plane and then extrude the sketch in a positive direction, the sketch plane is the BACK, or rear, face and the new extrusion face is the FRONT face.

You can also orbit using the ViewCube. Hold down the left mouse button on a cube corner, edge, or face, and drag the orbit icon to orbit freely. The display snaps to isometric, orthographic, and oblique views as you orbit. Right-click on the ViewCube or select the **Context Menu** flyout to display a menu of ViewCube settings and options. **Figure 6-6** describes ViewCube options.

NOTE

The ViewCube is customizable using the **ViewCube Options** dialog box. Use the ViewCube shortcut menu or select **ViewCube...** on the **Display** tab of the **Application Options** dialog box to display the **ViewCube Options** dialog box.

Figure 6-6.
Options available when you right-click on the ViewCube or select the **Context Menu** flyout.

Option	Description
Go Home	Displays the home view.
Orthographic	Activates the Orthographic camera view.
Perspective	Activates the Perspective camera view.
Perspective with Ortho Faces	Presents 3D views in perspective format and 2D views (looking at faces) in orthographic format.
Lock to Current Selection	When checked, display does not zoom in on a selected object.
Set Current View as Home	**Fixed Distance** option sets the current view, including zoom level, as the new home view. **Fit to View** sets the current rotation as the new home view and zooms the display to fit the graphics window.
Set Current View as Front	Redefines the current view as the front view.
Reset Front	Returns the front view to the original display.
Options...	Displays the **ViewCube Options** dialog box.

Using the View Face Tool

Use the **View Face** tool to rotate the display to a more convenient view. Select a face or plane to move the face or plane perpendicular to your line of sight. Select a line or edge to display the line or edge horizontal on-screen. See **Figure 6-7**.

> **NOTE**
>
> The **Previous View** and **Next View** tools are general view tools. The **Previous View** tool returns the view to the previous display, and the **Next View** tool advances to the display shown before you used the **Previous View** tool.

Exercise 6-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 6-3.

SteeringWheels

SteeringWheels provide unique navigation tools and an alternative means of using some view tools. Individual **SteeringWheels** are known as *navigation wheels*. The **Full Navigation Wheel**, shown in **Figure 6-8**, appears by default in modeling environments, and the **2D Navigation Wheel** is available in drawing files.

Navigation Bar

View Face

Ribbon
View
> Navigate

View Face

Ribbon
View
> Navigate

Previous View

Shortcut
Previous View

Type
[F5]

Ribbon
View
> Navigate

Next View

Shortcut
Next View

Type
[Shift]+[F5]

SteeringWheels: Circular navigation tools that allow you to navigate around a model.

Figure 6-7.
Use the **View Face** tool to rotate the view based on a selected plane, line, or edge.

Pick a plane or face...

...and the model rotates so that the face is perpendicular to your line of sight

...and the model rotates so that the line or edge is horizontal

Pick a line or edge...

Figure 6-8.
The **Full Navigation Wheel** provides access to many navigation tools. Wedges house each tool, similar to tool buttons.

Active wedge

Pick to close

Pick to display options
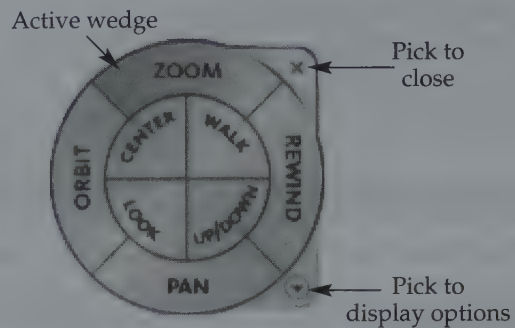
Navigation wheels are displayed next to the cursor. *Wedges* divide a navigation wheel and provide access to navigation tools, similar to tool buttons. Hover over a wedge to highlight it. You can pick some wedges to activate a tool directly. Others require that you hold down the left mouse button to use the tool. A navigation wheel remains on-screen until closed, allowing you to use multiple navigation tools. To close a navigation wheel, pick the **Close** button in the upper-right corner of the wheel, press [Esc], or right-click and pick **Close Wheel**.

NOTE

Most navigation tools function in model space. As a result, the location you zoom from or to, or orbit about, references a location on the model, not the center of the graphics window.

Zoom Navigation Tool

The **Zoom** navigation tool offers realtime zooming. Press and hold the left mouse button on the **ZOOM** wedge to display the pivot point icon and zoom navigation cursor. See **Figure 6-9**. The pivot point is the location at which you press the **ZOOM** wedge. Move the cursor toward the model to zoom in and away from the model to zoom out. The pivot point icon also zooms in or out as a visual aid. When you achieve the appropriate display, release the left mouse button.

Pan Navigation Tool

The **Pan** navigation tool offers realtime panning. Press and hold the left mouse button on the **PAN** wedge to display the pan navigation cursor. Move the cursor to pan and release the mouse button when you achieve the desired display.

Up/Down Navigation Tool

The **Up/Down** navigation tool allows you to pan the display along an axis perpendicular to the TOP and BOTTOM faces of the ViewCube, as if you are moving up or down in elevation in the real world. Press and hold the left mouse button on the **UP/DOWN** wedge to display the **Vertical Distance** indicator. While still holding down the button, move

Figure 6-9.
To use the **Zoom** navigation tool, move the cursor toward objects to zoom in and away from objects to zoom out.

Pivot point is located at the point in the graphics window where you press and hold the **Zoom** wedge
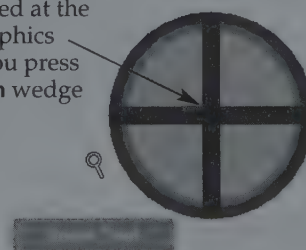
Figure 6-10.
Use the **Up/Down**
navigation tool to
adjust the view
elevation.

ViewCube

BOTTOM face

Drag up or down to
increase or decrease
the elevation
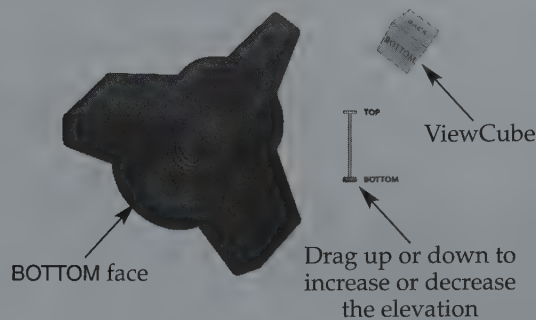
the cursor up or down to adjust the elevation. Release the button when you achieve the appropriate elevation. The link between the **Up/Down** tool and the ViewCube is most evident when you rotate to a view that does not show the TOP and BOTTOM faces in a horizontal orientation, as shown in **Figure 6-10**.

Center Navigation Tool

Use the **Center** navigation tool to pan the display from a selected point to the center of the graphics window. Press and hold the left mouse button on the **CENTER** wedge. The pivot point icon appears when you move the cursor over an object and snaps to points and edges. Release the mouse button to relocate the pivot point to the center of the graphics window. See **Figure 6-11**.

> **NOTE**
>
> The **Center** navigation tool also defines the pivot point used in the **Constrained** mode of the **Orbit** tool.

Orbit Navigation Tool

The **Orbit** navigation tool offers constrained orbit at the pivot point. Press and hold the left mouse button on the **ORBIT** wedge to display the pivot point icon and orbit navigation cursor. Move the cursor to orbit about the pivot point. To change the pivot point, while still holding the left mouse button on the **ORBIT** wedge, press [Ctrl] at the new pivot point.

Rewind Navigation Tool

The **Rewind** navigation tool allows you to observe the effects view tools have had on the drawing display, and return to a previous display. For example, if you use the navigation wheel to zoom in, then pan, then zoom out, you can rewind through each action and return to the original display, the zoomed-in view, the panned display, and then back to the current zoomed-out view. By default, you can rewind through view actions created using most view tools, not just those accessed from a navigation wheel.

Figure 6-11.
To use the **Center**
navigation tool,
move the cursor
over an object at the
point to center in the
graphics window
and release the left
mouse button.

Pivot icon appears only
when you move the cursor
over an object

This point will pan to
the exact center of the
graphics window
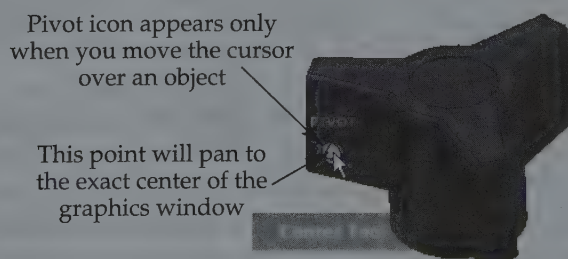
Figure 6-12.
Use the **Rewind** navigation tool to restore previous display configurations.

Orange frame indicates the current display

Move the brackets left and right to rewind in real time

Framed thumbnail view images
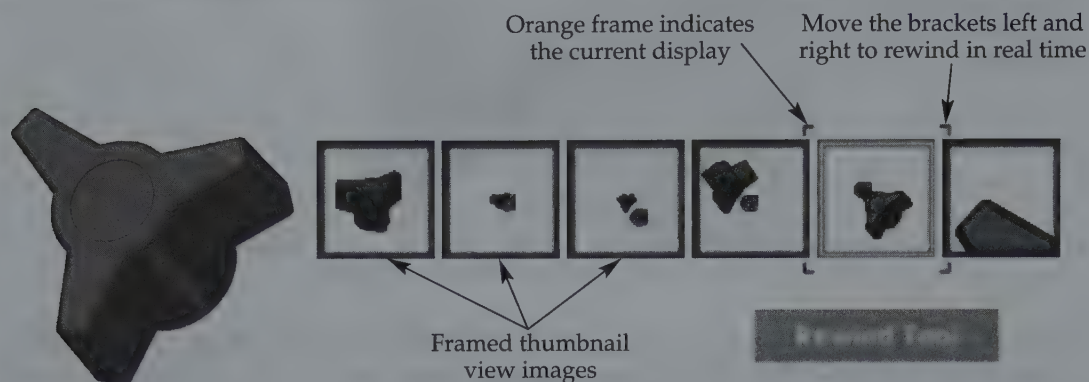
Pick the **REWIND** wedge once to return to the previous display. Thumbnail images appear in frames as the previous view restores. The orange-framed thumbnail surrounded by brackets indicates the restored display and its location in the sequence of events. See **Figure 6-12**. Pick the **REWIND** wedge repeatedly to cycle back through prior views. Another option is to press and hold the left mouse button on the **REWIND** wedge to display the framed view thumbnails. Then, while still holding the left mouse button, move the brackets left over the thumbnails to cycle through earlier views, or move the brackets right to return to later views. Release the mouse button when you achieve the desired display.

NOTE

The **Look**, **Walk**, and **Forward** navigation tools use the Perspective projection, described later in this chapter.

Navigation Wheel Options

A shortcut menu of options appears when you right-click on a navigation wheel or when you pick the flyout in the lower-right corner of a navigation wheel. The options vary depending on the current work environment. The first six options allow you to activate an alternative navigation wheel, as described in **Figure 6-13A**. The other options, which control general steering wheel characteristics or activate other tools, are described in **Figure 6-13B**.

Ribbon
View
> Appearance
Shaded

Ribbon
View
> Appearance
Hidden Edge

Ribbon
View
> Appearance
Wireframe

Exercise 6-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 6-4.

Shading

wireframe representation: A display with surfaces hidden, allowing you to see edges clearly.

Display, or shading, options allow you to view a model in a shaded, hidden-edge shaded, or wireframe format. See **Figure 6-14**. The **Shaded** mode is active by default and displays models in a shaded format. The **Shaded** mode looks realistic and allows you to observe a model as if it were an actual product. Activate the **Hidden Edge** mode to present models in a shaded format, but include hidden feature edges. Select the **Wireframe** mode to display models in a *wireframe representation*. A wireframe display removes all colors and lighting and allows you to view only feature edges. This can be helpful

Figure 6-13.
A—Available navigation wheels. B—Options for controlling navigation wheels.

Option	Description	Wheel
Mini View Object Wheel	Zoom, Rewind, Pan, and Orbit tools in mini format.	
Mini Tour Building Wheel	Walk, Rewind, Up/Down, and Look tools in mini format.	
Mini Full Navigation Wheel	Walk, Rewind, Pan, Orbit, Walk, Up/Down, Look, and Center tools in mini format.	
Full Navigation Wheel	Full navigation wheel tools in standard format.	
Basic View Object Wheel	Center, Zoom, Rewind, and Orbit tools in standard format.	
Basic Tour Building Wheel	Forward, Look, Rewind, and Up/Down tools in standard format.	

A

Option	Description
Go Home	Displays the home view.
Fit to Window	Activates the Zoom All tool.
Restore Original Center	Returns the pivot point to its original location.
Options...	Displays the SteeringWheel Settings dialog box, which is used to adjust steering wheel settings and to customize navigation wheels.
Close Wheel	Closes the navigation wheel.

B

Figure 6-14.
Adjust the model display as needed to view shaded surfaces or hidden edges, or to remove surfaces to analyze edges in a wireframe display.

Shaded   Hidden Edge   Wireframe

Ribbon
View
> Appearance

Orthographic

Ribbon
View
> Appearance

Perspective

Ribbon
View
> Appearance

No Shadow

Ribbon
View
> Appearance

Ground Shadow

Ribbon
View
> Appearance

X-Ray Shadow
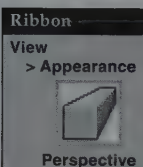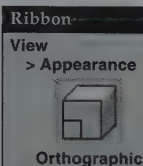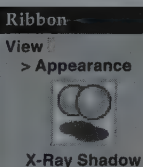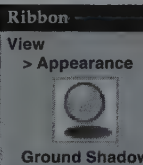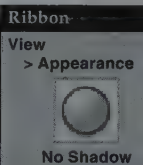
when you are working with complex features. The wireframe display format does not convert the model to a *wireframe model*.

Projection

The projection setting determines whether *projection foreshortening* applies to the model display. See **Figure 6-15**. The **Orthographic** mode, which is active by default, displays models without projection foreshortening. Features appear in a format that shows their true size and shape, allowing you to observe the actual dimensions of geometry.

Set **Perspective** mode to apply projection foreshortening. The **Perspective** mode displays the model closer to how the human eye would see a real object. While observing an object in **Perspective** view, use the view tools to apply a *fly-through* or *walk-through* effect. When you change the position of the object using view tools, the view depth and distortion also changes to reflect the new position.

NOTE

Shading and projection options can also be displayed in the navigation bar. To display them, pick the arrow at the bottom of the navigation bar to display the options, and select **Projection** and **Shading**.

Supplemental Material

Perspective View Navigation Tools
For information about navigation tools that are best used or only available in the **Perspective** view mode, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Perspective View Navigation Tools**.

Ground Shadow

Add ground shadow to the display to make a model look as if it is floating above the ground. The default **No Shadow** mode does not project a shadow. Select the **Ground Shadow** mode to project a shadow of the entire model. Choose the **X-Ray Shadow** mode to display a ground shadow similar to the **Ground Shadow** option, but with the addition of model edges. **Figure 6-16** shows each shadow option.

Figure 6-15.
Select a projection mode appropriate for your design requirements. For typical modeling, **Orthographic** mode is best to study the true size and shape of model geometry.

Orthographic

Perspective

Figure 6-16.
Add a shadow to create the effect of ground under the model and to help visualize and interpret a design.

No Shadow

Ground Shadow

X-Ray Shadow

NOTE

The **Display** tab of the **Application Options** dialog box provides options for adjusting wireframe and shaded display characteristics, as well as other model appearance settings.

Exercise 6-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 6-5.

Additional View Tools

Inventor includes many other view tools, many of them specific to file type or work environment. You will learn about some of these tools when applicable. Other tools, including those that allow you to display a file in multiple windows, clean the screen, or hide content are helpful mainly when working with large, complex models or drawings.

To display the active document in an additional window, pick the **New window** button from the **Windows** panel of the **View** ribbon tab. An additional window can be helpful by allowing you to display specific views in different windows. You can switch between windows as needed to work on the model.

The Inventor window can become crowded in the course of a design session. As the graphics window gets smaller, less of the model or drawing is visible. This can make drafting difficult. The **Clean Screen** tool toggles the clean screen display on and off. Cleaning the screen maximizes the size of the graphics window by clearing items such as the browser and displaying the ribbon as tabs only.

Ribbon
View
> Windows
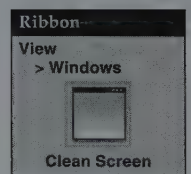
Clean Screen

PROFESSIONAL TIP

The **Clean Screen** tool can be helpful when you have multiple documents open. Only the active document appears when you use the **Clean Screen** tool. This allows you to work more efficiently within one of the files.

The **Object Visibility** flyout in the **Visibility** panel of the **View** ribbon tab controls the visibility of sketches and categories of work features. These selections are useful for managing a cluttered graphics window.

Model Style Fundamentals

Styles in the model work environment specify color, lighting, and material characteristics. For example, if you build a model of a stainless steel part, you can use a stainless steel color, a specific lighting configuration, and a stainless steel material. Numerous default styles are available that you can assign to a model. If you cannot find an existing style appropriate for your application, you can create your own style.

Material Style

Inventor provides a large number of material styles, ranging from lead to ultra-high molecular weight polyethylene. Assign a material style to a model to create a virtual representation of the real product material. Material styles include properties that influence the physical and inertial characteristics of the model. Each material style also references a color style for correct model appearance. Additionally, material is a model property that you can reference throughout the design and documentation process in items such as a parts list, bill of materials, or drawing title block.

Ribbon
**Manage
> Styles and
Standards**

Styles Editor

Select a material from the **Material** drop-down list on the **Physical** tab of the **iProperties** dialog box, or from the **Material** drop-down list on the **Standard** tab of the **Document Settings** dialog box. You can also assign a material from the **Style and Standard Editor**. Expand the **Material** node and double-click on a material to activate. Pick a specific material to display material settings. **Figure 6-17** shows methods of activating a material style. This example uses a Steel, High Strength Low Alloy material that references a Metal-Steel(Polish) default color style.
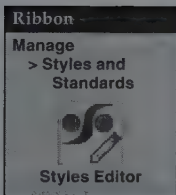
Color Style

By default, a material style references a color style to display a model using a color appropriate for the specified material. For example, the Gold material references a Metal-Gold color to make the model appear as if it is made of gold. Color selection varies depending on the model and the design requirements. Use the Default material and color throughout the design process if the exact material is not important and a grey color is acceptable. A more common option is to assign a material to the part and allow the material to specify the color, as previously described. You can also *override* the material color to simulate painting or color added to the material.

override: Make a temporary change to the current style settings.

Quick Access

As Material
Color Override

Ribbon
**View
> Appearance**

As Material
Color Override

To override the model color, select a color from the **Color Override** flyout. You can also specify a color override from the **Style and Standard Editor**. Expand the **Color** node and double-click on a color to activate. Pick a specific color to display color settings. **Figure 6-18** shows methods of activating a color style. This example uses a Chrome color override to represent a chrome finish, or appearance. Many different colors are available, including some with texture options.

Figure 6-17.
A—Use the **iProperties** dialog box to specify model properties referenced throughout design and documentation, including the material. B—You can also use the **Document Settings** dialog box to activate a material style. C—The **Style and Standard Editor** allows you to activate and manage styles.

Select a material from the list

Selected material is assigned a specific color

Model physical properties with the selected material applied

A

Select a material from the list

B

Double-click to activate; active material is bold

Material style settings
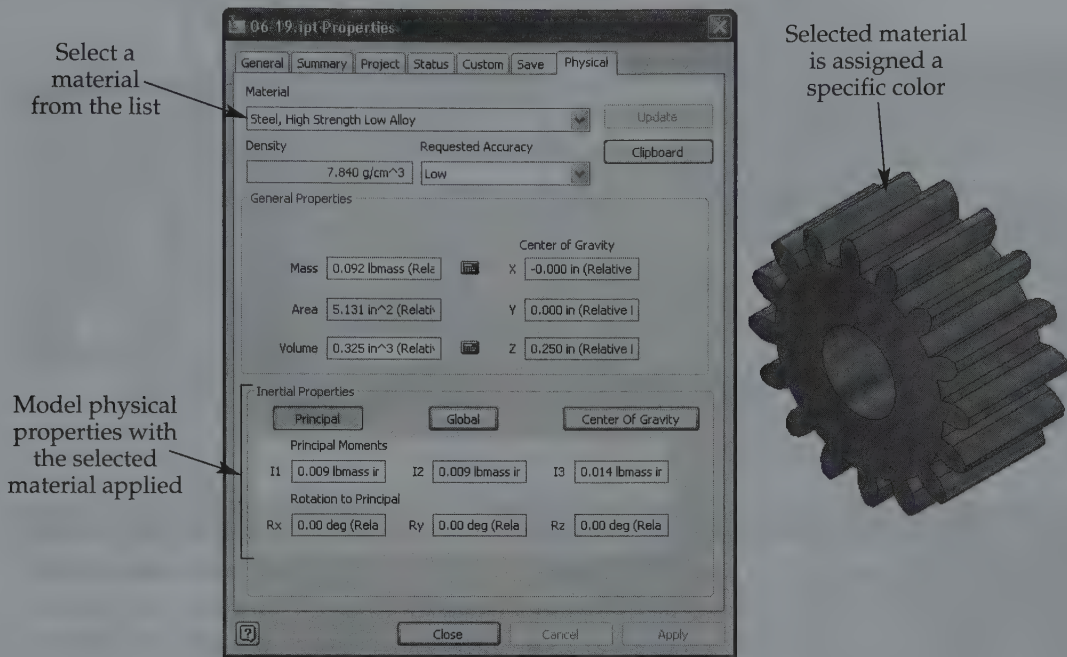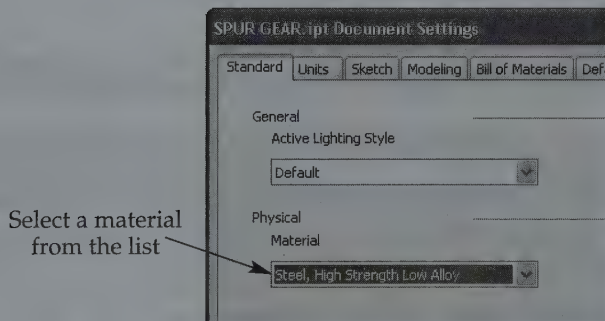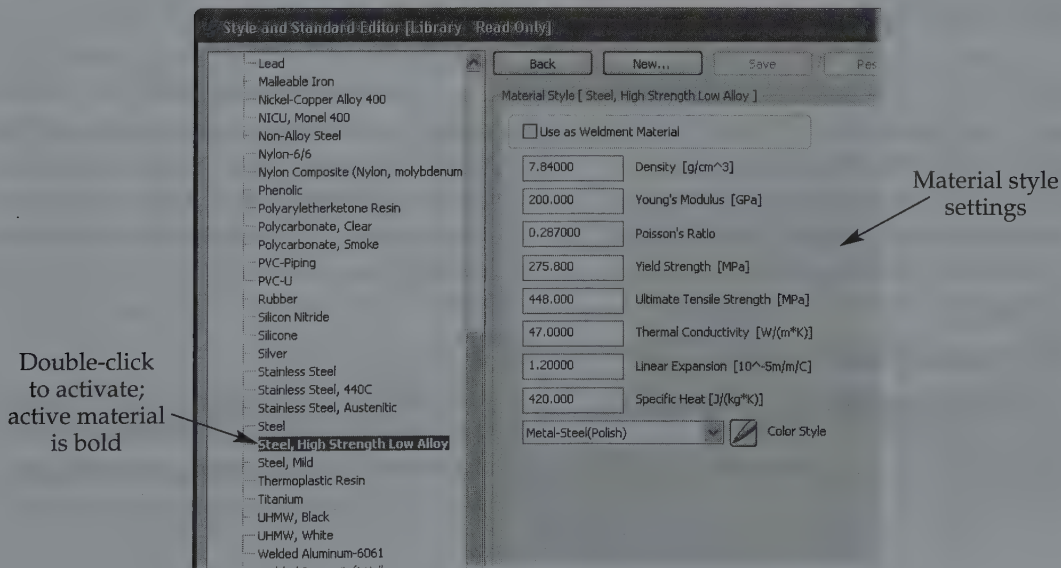
C

Figure 6-18.
A—The **Color Override** flyout is available from the **Quick Access** toolbar or **Appearance** panel of the **View** ribbon tab, and is the fastest way to apply a unique color to the model. B—The **Style and Standard Editor** allows you to activate and manage color styles.

A   B
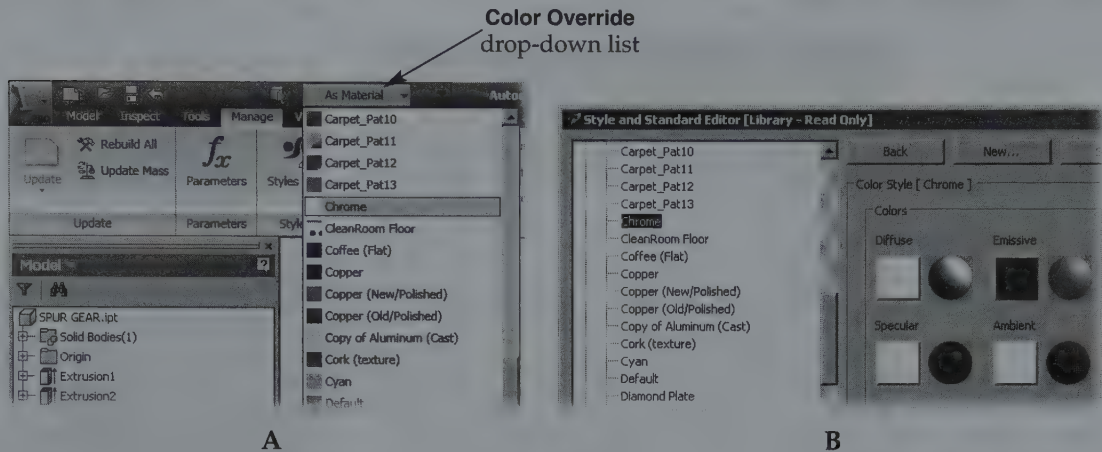
> **NOTE**
>
> The project **Use Style Library** option must be set to **Yes** or **Read Only** in order to display styles in the style library. You may also need to pick **All Styles** from the **Filter** drop-down list in the **Style and Standard Editor** to display styles in the style library.

> **PROFESSIONAL TIP**
>
> You can specify a specific material style for a part, but use a different color in order to distinguish parts in an assembly, or to simulate painting or color treatment. However, to use the color referenced by the selected material, be sure to use the As Material color option, which does not override the material color.

Lighting Style

Lighting style defines various lighting characteristics applied to the model for appearance purposes. Select a lighting style from the **Active Lighting Style** drop-down list of the **Document Settings** dialog box. You can also activate a lighting style in the **Style and Standard Editor**. Expand the **Lighting** node and double-click on the style to activate it. Pick a specific lighting style to display lighting settings. **Figure 6-19** shows methods of activating a lighting style. This example uses the Two Lights lighting style.

Supplemental Material   *Creating and Saving Model Styles*

For information about developing color, material, and lighting styles, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Creating and Saving Model Styles**.

Figure 6-19.
A—The **Document Settings** dialog box provides a convenient way to activate a lighting style.
B—The **Style and Standard Editor** allows you to activate and manage lighting styles.

Select a lighting style from the list

A

Double-click to activate; active lighting is bold

Lighting style settings

B
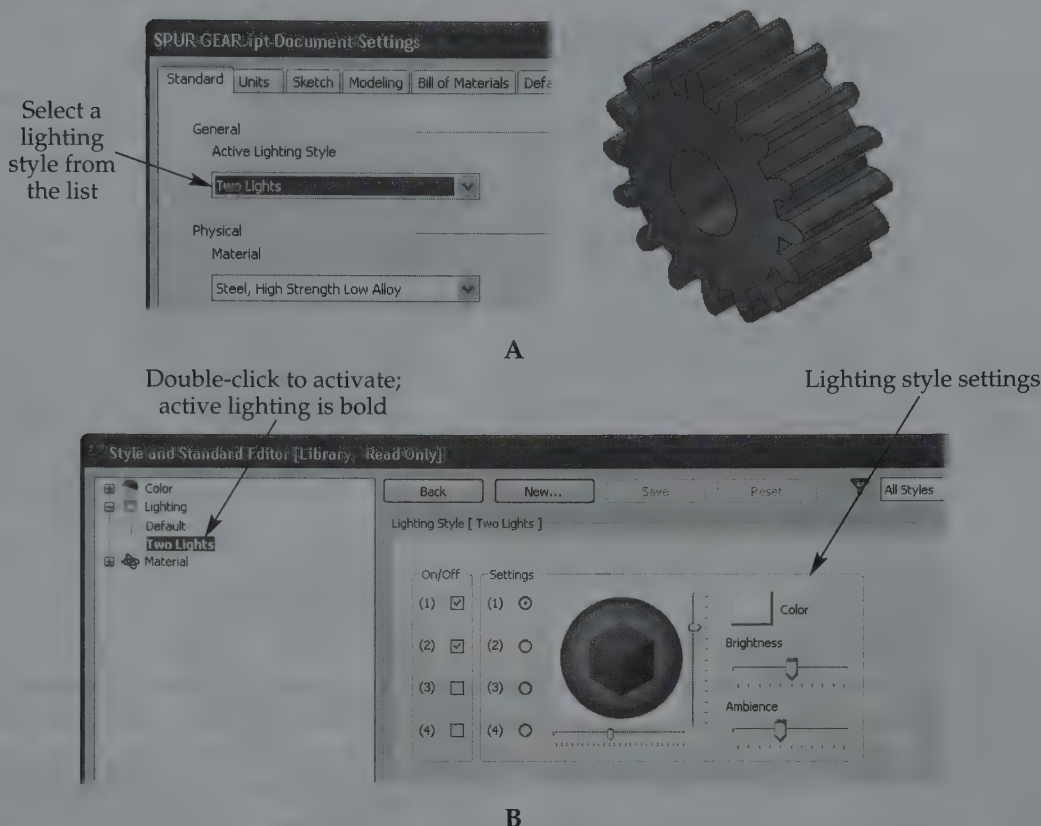
Basic Model Inspection

Inventor provides a number of tools for viewing and analyzing model characteristics and properties. This chapter focuses on basic tools and options used to view and measure a model. Use measurement tools frequently throughout the design process to inspect model geometry and assist in the design process. Many other, more advanced model analysis tools are also available.

Physical Properties

Use the **Physical** tab of the **iProperties** dialog box to assign a material to a model and view how the selected material affects the physical and inertial properties of the model. See **Figure 6-20.** Select an accuracy level from the **Requested Accuracy** list. If you make changes to the mass properties of the model, such as adding or removing material, you must update the physical properties to display the correct calculations. Pick the **Update** button to update the physical properties. Pick the **Clipboard** button to copy the properties for pasting into a document or spreadsheet file.

> **NOTE**
>
> You can also update mass properties by selecting the **Update Mass** button of the **Update** panel in the **Manage** ribbon tab.

Figure 6-20.
Use the **Physical** tab of the **iProperties** dialog box to assign a material to the model and view the effects of a specific material on the physical and inertial properties of the model.

Pick to copy properties to the clipboard

Physical and inertial properties of the model are calculated based on the selected materia

Model resembles selected material
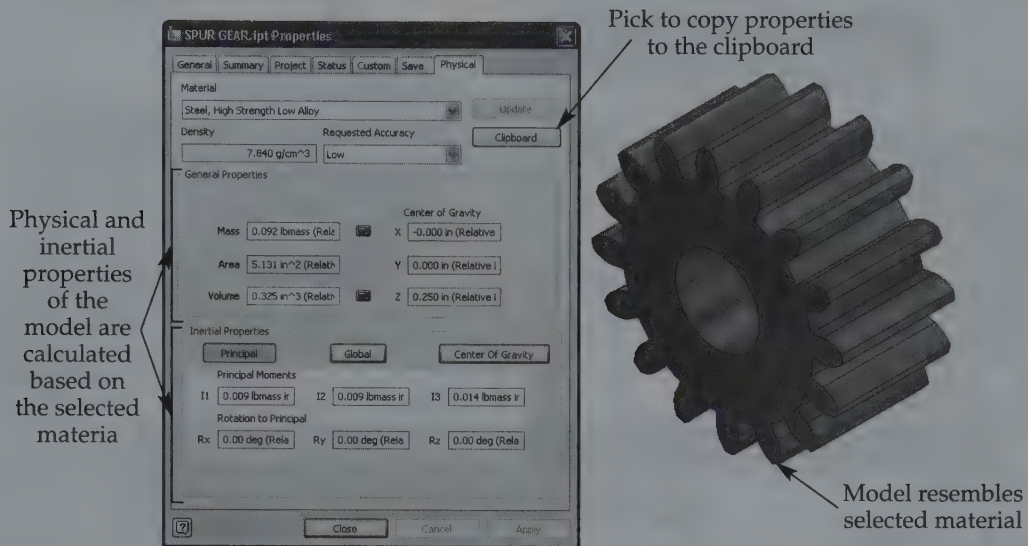
Center of Gravity

center of gravity: The center of model mass, where balance occurs.

Ribbon

View
> Visibility
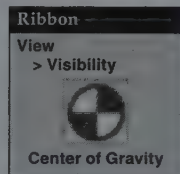
Center of Gravity

Use the **Center of Gravity** tool to toggle the display of the model's *center of gravity*. See **Figure 6-21**. The center of gravity appears as the Center of Gravity triad, which includes points, axes, and planes. You can use the **Center of Gravity** tool when developing basic part models, but is often more useful for working in an assembly. In an assembly, the **Center of Gravity** tool helps you analyze the center of gravity of a component in relation to the assembly and other components. Select the large point, or ball, to display the coordinates of the center of gravity from the XYZ center point of 3D space. Select the small work point or work planes for reference purposes, such as for taking measurements. The axes and arrowheads appear for reference and help you to confirm the position of the model in 3D space.

Figure 6-21.
Display the Center of Gravity triad to identify the model center of gravity and to take measurements from the center of gravity.

Pick to display the coordinates of the center of gravity

Center of gravity coordinates

CG: -0.00 in, 0.00 in, 0.25 in
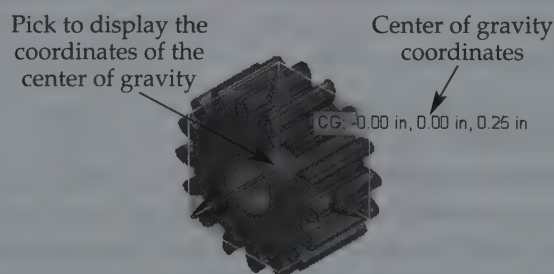
Taking Measurements

Use the **Measure Distance** tool and dialog box to take distance measurements. See **Figure 6-22A**. You can pick most objects to measure, such as the length of an edge, diameter of a circle, or radius of an arc. You can also measure the distance between two objects, including points, curves, edges, faces, or planes. Note that when you measure from a point to a plane, the distance is measured perpendicular to the plane. Typically, the **Measure Distance** tool calculates the most direct distance.

Use the **Measure Angle** tool and dialog box to identify the *included angle* between two objects, including the angle between a curve and a point. See **Figure 6-22B**. The **Measure Loop** tool measures the distance around, or perimeter of, a selected closed loop. See **Figure 6-22C**. Typically, use the **Measure Area** tool to measure the area of a surface. See **Figure 6-22D**.

Ribbon
Tools
> Measure

Measure Distance
Type
[M]
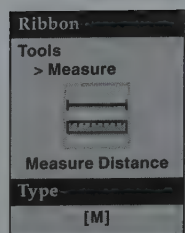
> **included angle:** The angle between two selected edges, curves, axes, faces, planes, or a combination of objects, such as an edge and a face.

Ribbon
Tools
> Measure
Inspect
> Measure

Measure Angle

Ribbon
Tools
> Measure
Inspect
> Measure

Measure Loop

Ribbon
Tools
> Measure
Inspect
> Measure

Measure Area

Figure 6-22.
A—Using the **Measure Distance** tool to measure the length of an edge. B—Using the **Measure Angle** tool to measure an included angle between edges. C—Using the **Measure Loop** tool to measure the perimeter of a loop. D—Using the **Measure Area** tool to measure the area of a surface.

Selected edge

Measure cursor icon

Measure Distance
0 in
Length
3 in

Information and display vary depending on whether you pick points, edges, or planes

A

Measure Angle
120 deg

Angle appears in display box

Selected edges

B

Selected loop

Line Length
15.5 in

Loop length appears in display box

C

Selected face

Area
9.417 in^2

Area appears in display box

D
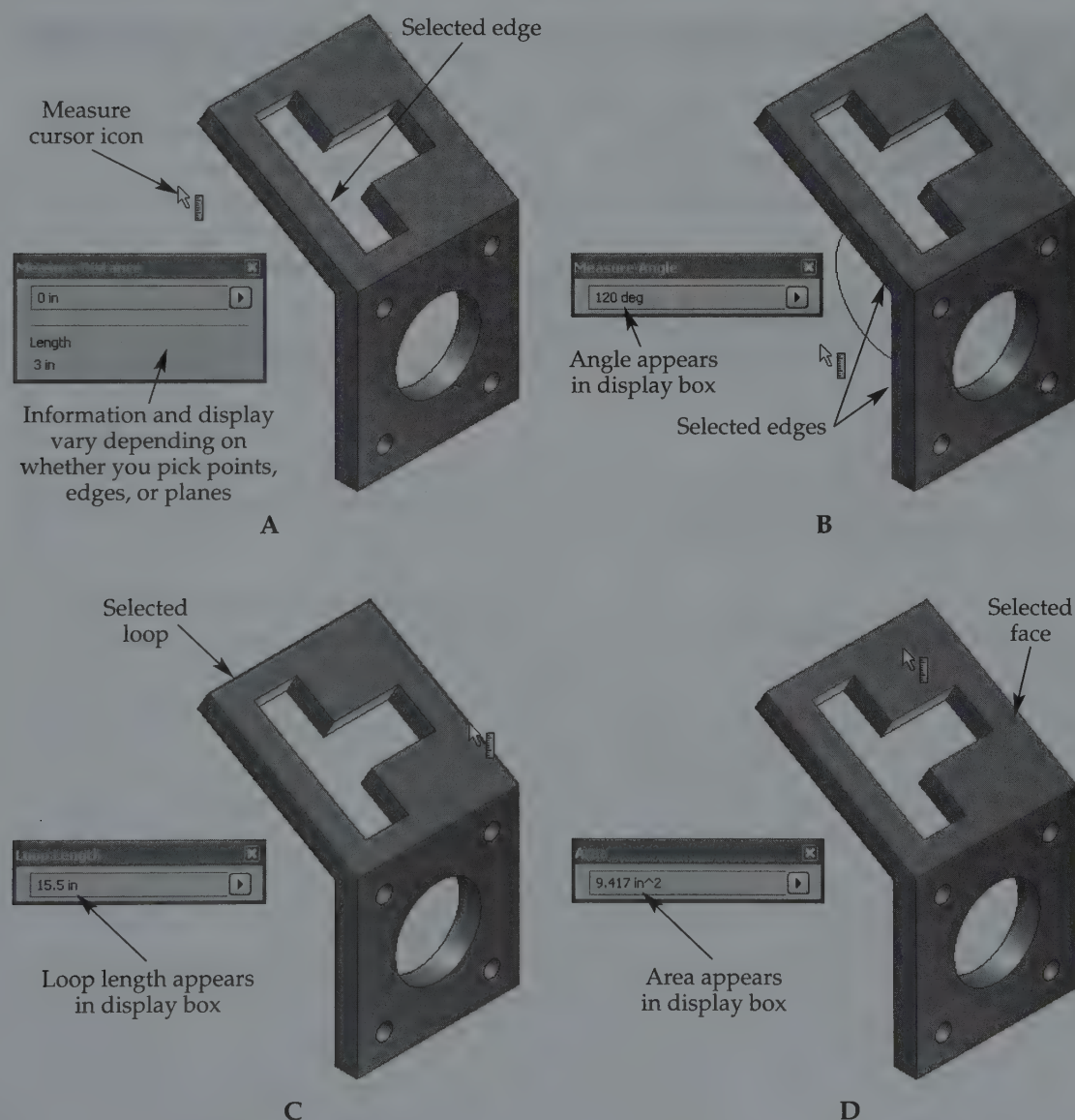
Each measurement dialog box includes a flyout with options for setting measurement preferences, selecting a different tool, and accumulating values. Before taking a measurement, select the appropriate precision from the **Precision** cascading menu. The default precision is the precision assigned to document units. This can sometimes be misleading. Pick a higher precision or select the **All Decimals** option to return the most accurate value. To display the value using the document units as well as alternate units, select an option other than **None** from the **Dual Unit** cascading menu.

Once you take a measurement, pick the **Restart** option to take a fresh measurement. The **Measure Distance**, **Measure Angle**, **Measure Loop**, and **Measure Area** options activate the appropriate tools. Use the **Add to Accumulate**, **Clear Accumulate**, and **Display Accumulate** options to add, or accumulate, multiple distances or angles.

> **NOTE**
>
> A flyout with options for selecting entire components, parts, or part features may appear while you are taking measurements, allowing you to define the items you select while working on an assembly.

> **PROFESSIONAL TIP**
>
> Use the **Rebuild All** tool to ensure that all model geometry is calculated and accounted for, especially when you are working with large, complex models.

Ribbon
Manage
  > Update

Rebuild All

Exercise 6-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 6-6.

Chapter Test

1. Describe the changes view tools make to the physical location, size, and shape of objects.
2. What is the difference between screen space and model space as related to using view tools?
3. What do zoom tools allow you to do?
4. Name the default mode of the **Orbit** tool.
5. What is the pivot point?
6. What does the ViewCube allow you to do?
7. Define *orthographic view*.
8. Briefly describe an isometric view.
9. What is an oblique view?
10. Describe the basic function of the **View Face** tool.
11. What is another name for individual SteeringWheels?
12. What are wedges, and how do they function?
13. In the **Rewind** tool, what does the orange-framed thumbnail surrounded by brackets indicate?
14. Describe the characteristics of a model in the **Shaded** display mode.
15. Which display option displays a model in a shaded format but includes hidden feature edges?
16. Briefly describe the appearance of models in a wireframe format, and explain why this can be helpful.
17. Describe the appearance of models in the **Orthographic** view mode.
18. How are models displayed in the **Perspective** view mode?
19. Define *fly-through*.
20. Define *walk-through*.
21. Briefly describe the purpose of model styles, and give an example of their use.
22. Briefly describe the relationship between color styles and material styles.
23. What effect does assigning a material style to a model have on the properties of the model?
24. What is the center of gravity of a model?
25. What tool allows you to measure the included angle between two objects?

Problems

1–10 Instructions:
- *Open the specified part file, and save the file using the given name.*
- *In each file, activate the given material and color style.*

1. **File:** P4-5.ipt
   Save as: P6-1.ipt
   Title: SUPPORT BLOCK
   Material: Titanium
   Color: As Material
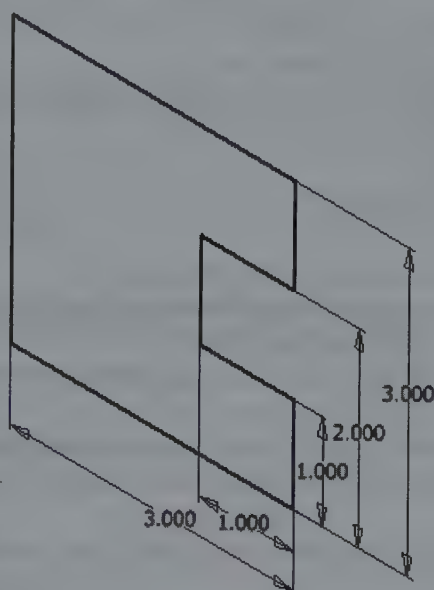
2. **File:** P5-1.ipt
   Save as: P6-2.ipt
   Title: PIN
   Material: Steel, High Strength Low Alloy
   Color: As Material

3. **File:** P5-2.ipt
   Save as: P6-3.ipt
   Title: SWIVEL
   Material: Steel, High Strength Low Alloy
   Color: As Material
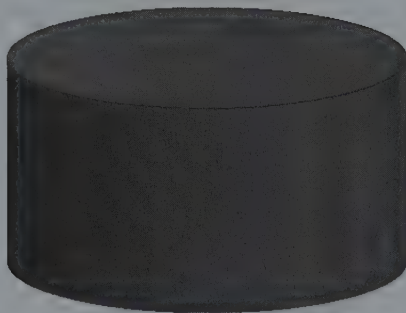
4. **File:** P5-3.ipt
   Save as: P6-4.ipt
   Title: SCREW
   Material: Steel, High Strength Low Alloy
   Color: As Material

5. **File:** P5-4.ipt
   Save as: P6-5.ipt
   Title: SUPPORT BRACKET
   Material: Aluminum-6061
   Color: Gold Metallic
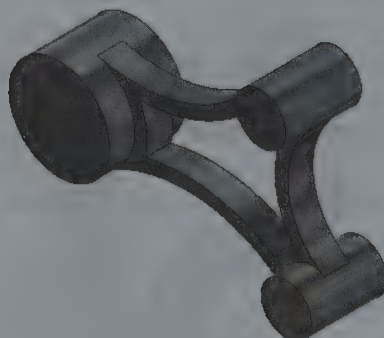
6. **File:** P5-5.ipt
   Save as: P6-6.ipt
   Title: BODY
   Material: Steel, High Strength Low Alloy
   Color: Red

7. **File:** P5-6.ipt
   Save as: P6-7.ipt
   Title: HEAD
   Material: ABS Plastic
   Color: Green

8. **File:** P5-7.ipt
   Save as: P6-8.ipt
   Title: HANDLE
   Material: Stainless Steel
   Color: As Material

9. **File:** P5-8.ipt
   Save as: P6-9.ipt
   Title: STOP
   Material: Stainless Steel, 440C
   Color: As Material
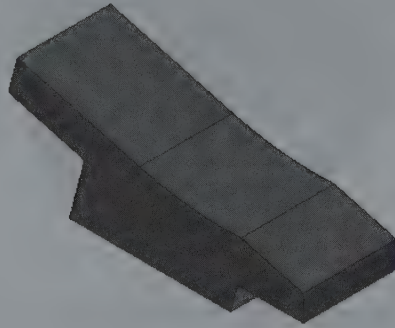
10. **File:** P5-9.ipt
    Save as: P6-10.ipt
    Title: FUNNEL
    Material: Acetal Resin, Black
    Color: As Material

11–14 Instructions: Follow the specific instructions for each problem.

11. Perform the following tasks:
    A. Open Rim.ipt from the following folder: Autodesk/Inventor 2010/Samples/Models/Parts/Rim. If the Exercises and Problems project is active, you should be able to locate this file quickly by picking the **Samples** folder in the **Locations** area of the **Open** dialog box.
    B. Save a copy of the Rim model as P6-11.ipt.
    C. In the P6-11.ipt file, use the **iProperties** dialog box to change the material to Cast Steel.
    D. Use the **Color** drop-down list on the **Inventor Standard** toolbar to change the color from Chrome to Yellow, and then to As Material.
    E. Access the **Style and Standard Editor** and activate the Aluminum-6061 material style.
    F. Pick the Aluminum-6061 material style if not already selected, and notice the Metal-AL-6061 (Flat) color style definition.
    G. Expand the **Color** node and notice that the active color style is Metal-AL-6061 (Flat).
    H. Activate the Black color style and notice the effect on the model.
    I. Return to the Aluminum-6061 style of the **Materials** node and notice that the material color is still Metal-AL-6061 (Flat).
    J. Resave P6-11.ipt.

Problems – Chapter 6

12. Create a freehand sketch of the standard **Full Navigation Wheel**. Label each of the wedges and provide a brief description of each tool.

13. Create a new part file using the Part-IN.ipt template from the **New File** dialog box. Perform the following tasks:
    A. Open a new sketch on the XY plane and sketch a 24″ × 48″ rectangle with the lower-left corner coincident to the projected center point.
    B. Extrude the sketch .0375″ in the positive direction.
    C. Access the **Style and Standard Editor** and select Metal-Steel from the **Color** node.
    D. Pick the **New…** button and change the name from Copy of Metal-Steel to Screen.
    E. Pick the **Choose** button and select the Screen_3& texture style.
    F. Change the **% Scale** to 400%.
    G. Save and close the **Style and Standard Editor**.
    H. Change the color of the part to Screen.
    I. Save the part as P6-13.ipt.
    The final part should look like the screen shown.
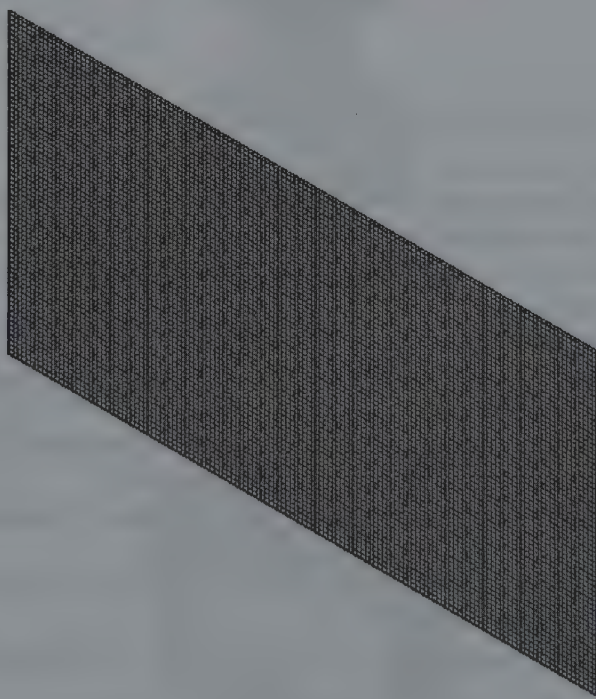
14. Open the blade_main.ipt sample part from the following path: Autodesk/Inventor 2010/Samples/Model/Assemblies/Scissor/Components/blade_main.ipt. Rotate the model to an isometric view and draw a freehand sketch of the part.

CHAPTER

Holes, Bends, Ribs, and Webs

7

Learning Objectives

After completing this chapter, you will be able to do the following:
✓ Add holes.
✓ Create bends.
✓ Develop ribs and webs.

Add holes, bends, ribs, and webs to replicate specific manufacturing processes or part structures. You can also use these and other features in a more general way to develop the shape of a part model. As you learn to create holes, bends, ribs, and webs, consider how they can apply to unique design requirements.

Holes

The **Hole** tool allows you to replicate a drill, cut, bore, core, or other hole operation. You can also create a hole whenever a cut cylinder is required, similar to a cut extrusion. However, unlike the **Extrude** tool, the **Hole** tool provides many specific hole options, such as common hole characteristics and threading. Access the **Hole** tool to create a hole feature using the **Hole** dialog box. See **Figure 7-1**.

Ribbon
Model
> Modify
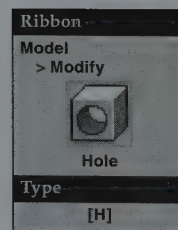
Hole
Type
[H]

Hole Placement

You can locate a hole using a sketch, existing feature edges, or a work point. Establish how and where to place holes before defining them. Then select a placement option from the **Placement** drop-down list.

Linear

Select the **Linear** placement option to locate a single hole on a feature using linear distances and without creating a separate sketch. The **Linear** placement method is the default if an unconsumed sketch with selectable points is not present. The **Face** button is active by default, allowing you to choose the planar feature face on which to begin the hole. See **Figure 7-2**.

Figure 7-1.
The **Hole** dialog box with the **Linear** placement, **Drilled**, **Angled** drill point, and **Simple Hole** options selected.

Select a hole type

Specify hole dimensions

Pick to create an angled drill point

Pick to create a flat drill point

Pick to create a hole without threads

Pick a threading option
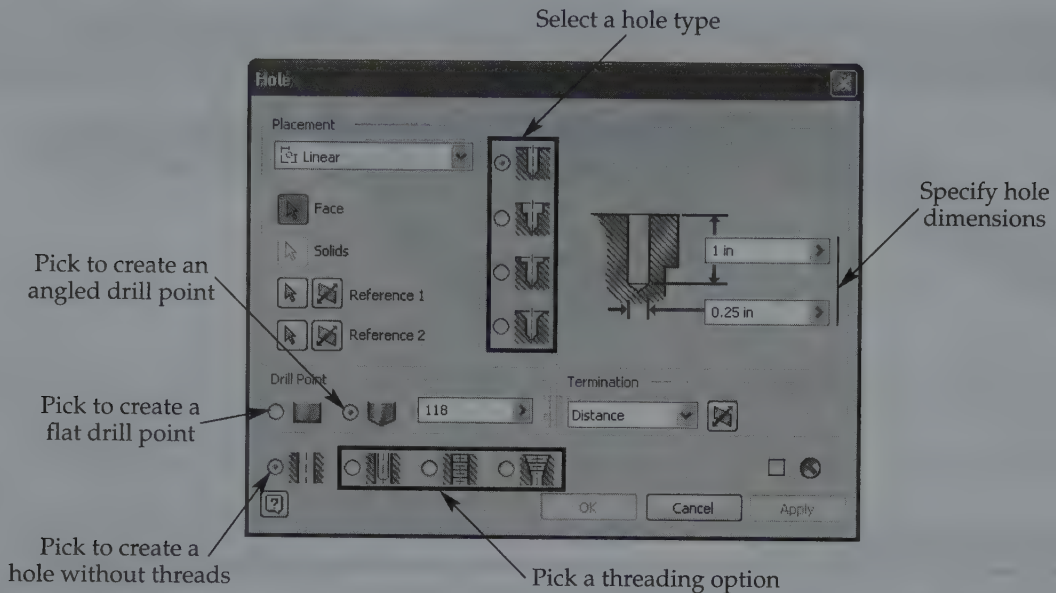
Figure 7-2.
Follow these steps to insert a hole using the **Linear** placement option.

Hole preview

First linear dimension

First selected edge

Second selected edge

Second linear dimension

Selected face

1. Select a face to place the hole

2. Select an edge and enter a distance for the first dimension

3. Select an edge and enter a distance for the second dimension
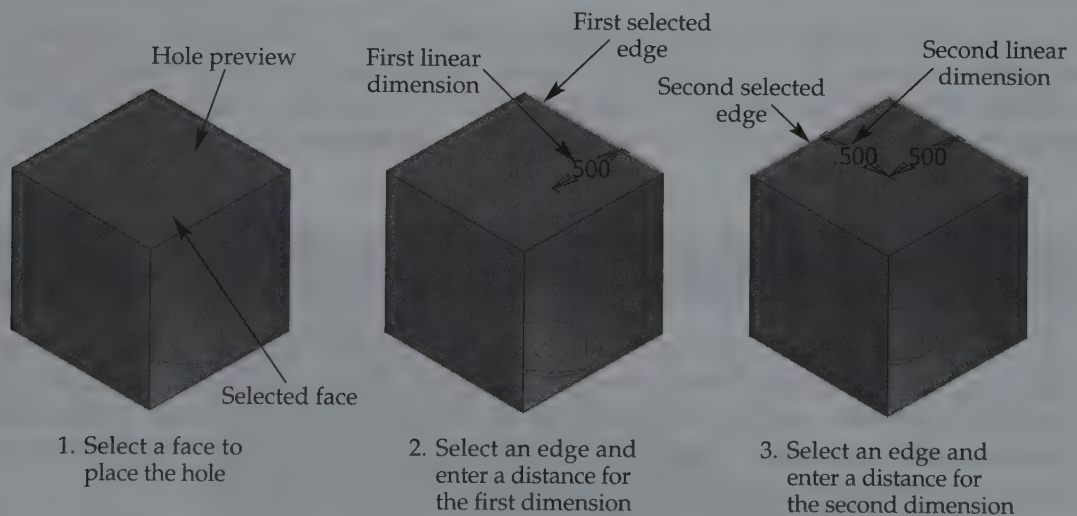
After you pick a face, the **Reference 1** select button activates, allowing you to pick an edge to define the first dimension to the center of the hole. If the **Edit dimension when created** function is set, the **Edit Dimension** dialog box appears, and you can specify the distance. If the **Edit dimension when created** function is not set, double-click on the dimension after you establish **Reference 2**, and modify the value. Once you define the first reference, the **Reference 2** select button activates, allowing you to select another edge to define the second dimension to the center of the hole. Use the **Edit Dimension** dialog box to specify the value. The hole is now precisely located on the selected face.

NOTE

If necessary, use the **Reference 1** and **Reference 2** flip buttons to reverse the side of the selected edges on which the dimension occurs.

From Sketch

The **From Sketch** placement option requires that you create a sketch on a face or plane before you access the **Hole** tool. Sketch points created using the **Point, Center Point** tool and **Center Point** format are common for locating the center of holes. Constrain the points to locate holes precisely. You can also use other sketch points, such as the endpoint of a curve or the center point of a fillet, arc, or circle. See **Figure 7-3**.

If the model includes an unconsumed sketch with an appropriate sketch point, the **From Sketch** option is active when you access the **Hole** tool. All points sketched using the **Point, Center Point** tool and **Center Point** format are selected automatically. To place holes at other available points, you must use the active **Centers** button to pick the points. When you place a hole using a sketch, the hole acts as a true sketched feature and consumes the sketch. Other options function more like placed features.

> **NOTE**
>
> To deselect points, pick the **Centers** button, hold down the [Ctrl] key, and choose the points to remove from the selection set.

> **PROFESSIONAL TIP**
>
> You can use a single sketch with multiple sketch points to create different types of holes from the same sketch. Create the first hole(s) as one hole type, and then right-click on the consumed sketch in the browser and select **Share Sketch** to share the sketch. Access the **Hole** tool and create additional holes with other characteristics, picking the **Apply** button after each hole. The dialog box remains open, allowing you to create a variety of holes quickly.

Concentric

Select the **Concentric** placement method to place a hole concentric to a cylindrical or conical feature without using a separate sketch. The **Plane** button activates, allowing you to choose the face on which to place the hole. The **Concentric Reference** select button then activates. Pick an appropriate cylindrical feature edge to locate the hole center concentric to the feature. See **Figure 7-4**.

Figure 7-3.
Using a sketch and the **From Sketch** placement option to locate the center of a hole.
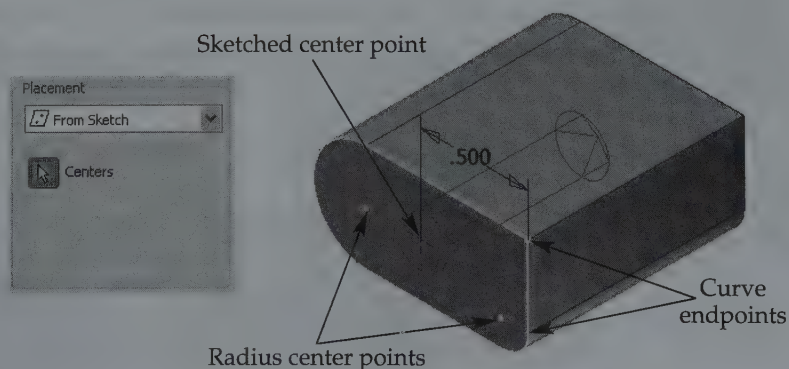
Sketched center point

.500

Curve endpoints

Radius center points

1. Select a plane or face \ Hole preview | 2. Pick the concentric reference \
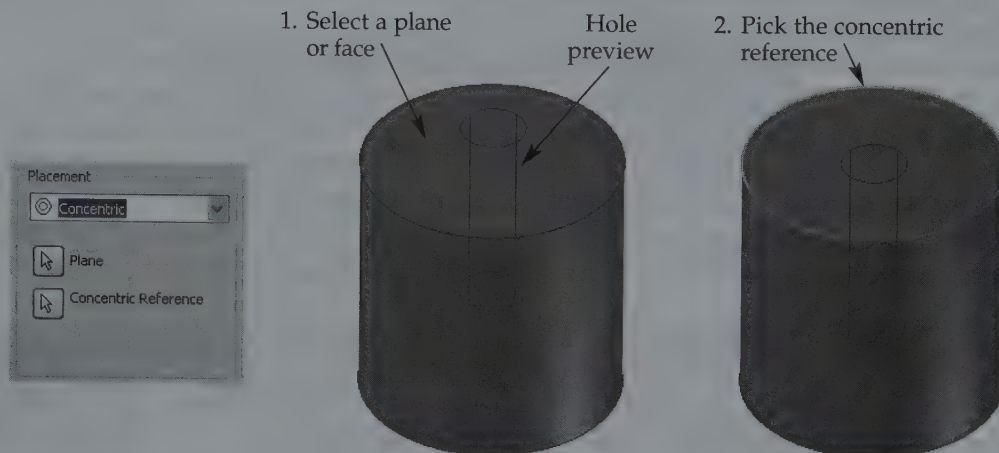
On Point

Choose the **On Point** option to position a hole at a work point. Chapter 11 describes work points, but in general, they allow you to create a reference point anywhere in space. The center point in the **Origin** folder of the browser is an example of a work point. When you access the **On Point** option, the **Point** button becomes selected, allowing you to choose a single existing work point or the center point from the browser. The **Direction** select button then activates, allowing you to select an object to define the direction of the hole axis. See **Figure 7-5**.

Hole Characteristics

The **Hole** tool allows you to add common hole characteristics to create a specific type of hole. See **Figure 7-6**. Pick the **Drilled** radio button to create a *drilled hole*. Select the **Counterbore** radio button to specify a *counterbored hole*. Choose the **Spotface** radio button to add a *spotface* at the top of the hole. Pick the **Countersink** radio button to specify a *countersunk hole*. Use the appropriate text boxes in the preview area to specify hole sizes.

> **NOTE**
>
> You can apply *threads* or fastener information to drilled, counterbored, spotface, and countersunk holes, but you cannot apply tapered threads to counterbored holes.

drilled hole: The most basic hole type, with no counterbore, spotface, or countersink where the hole begins.

counterbored hole: A drilled hole that has a larger-diameter, cylindrical opening at the top; typically used when a flush surface is necessary, such as to hide a binding screw head.

spotface: Similar to a counterbored hole, but shallower; typically applied when a flush surface is necessary, such as to hide a flat washer, and in casting applications.

countersunk hole: Similar to a counterbored hole, but the recess is tapered, resulting in a conical shape that is often used to hide a screw head.

threads: Spiral or helical grooves cut in or around the face of a cylindrical or conical feature.

Figure 7-5.
Using the **On Point** placement option.

Selected work point
Selected plane
Direction preview
Hole preview
Any straight object, face, or plane can be used to define the direction
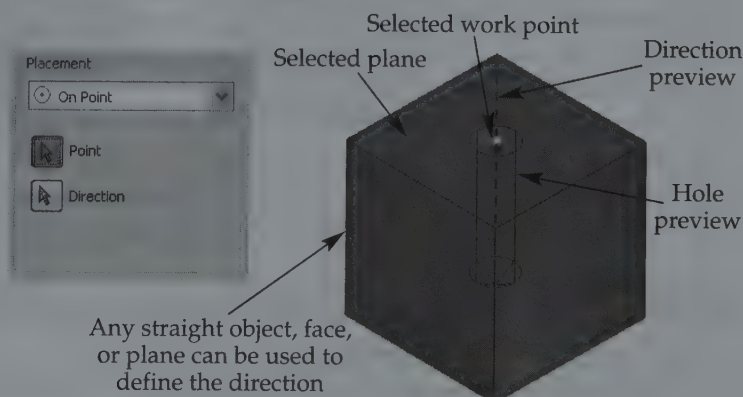
Figure 7-6.
Use the **Hole** tool to create a variety of common holes.

Drilled   Counterbored   Spotface   Countersink
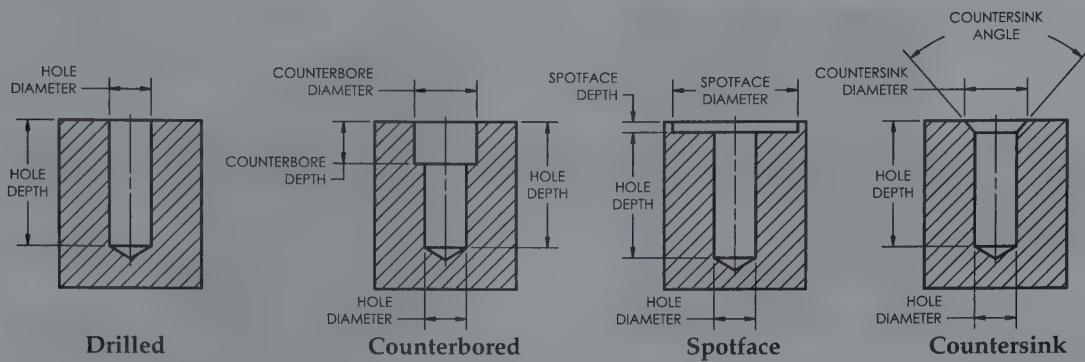
Termination and Drill Point

The **Termination** area allows you to control the depth of the hole. Use the **Distance** option to end a hole at the depth specified in the preview area. See **Figure 7-7A**. Select the **Through All** option to drill a hole through all features. See **Figure 7-7B**. Both options include a **Flip** button for reversing the direction of the hole if necessary. Use the **To** option to choose a face or plane to end the hole using the active **Select surface to end the feature creation** button. See **Figure 7-8**. If the face or plane you choose does not intersect the hole path, select the **Terminate feature on extended face** check box.

The **Drill Point** area allows you to define the drill point characteristics for holes that do not run through all material. Pick the **Flat** radio button to create a flat, 0° drill point. To form an angled drill point, pick the **Angle** radio button and specify an angle using the text box.

NOTE

Holes with certain termination or tapped options include fewer preview area text boxes because termination and thread parameters control size dimensions.

Figure 7-7.
A—Use the **Distance** termination option to specify the depth of a hole. B—Use the **Through All** option to create a thru hole.

Enter or select a depth

Pick to flip the direction

Hole cuts through all material

Pick to flip the direction

A   B
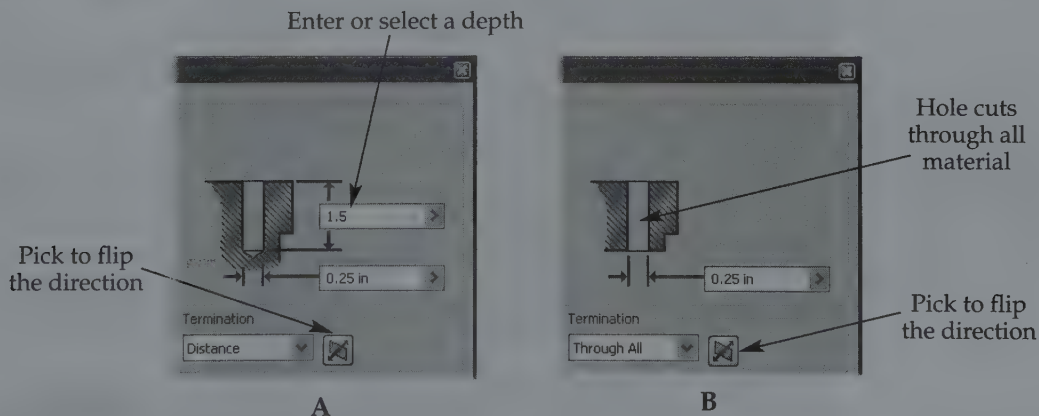
Creating a hole using the **To** termination option (part sectioned for clarity).

Hole begins at this face

Hole ends at this face
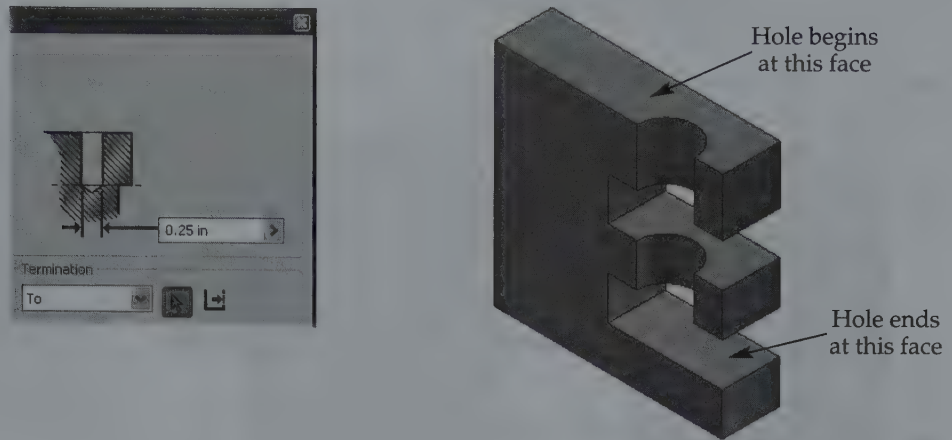
Threads

tap: Use a machine tool to form an interior thread.

The **Simple Hole** radio button, selected by default, allows you to create holes without threads or a specific fastener association. Pick the **Tapped Hole** radio button to *tap* or otherwise cut threads in holes using the options in the **Threads** area. See **Figure 7-9**. The **Full Depth** check box is selected by default and applies threads to the entire depth of the hole. To thread a portion of the hole, deselect the **Full Depth** check box and specify the thread depth using the corresponding text box in the preview area.

right-hand threads: Threads that move a right-hand threaded screw forward in a clockwise direction.

The **Thread Type** drop-down list allows you to specify the thread standard used to create the threaded hole. Choose ANSI Unified Screw Threads for inch parts, ANSI Metric M Profile for metric parts, or one of a variety of other thread standards, including ISO, DIN, BSP, and JIS. Use the **Direction** area to form *right-hand threads* or *left-hand threads* by choosing the appropriate radio button.

left-hand threads: Threads that move a left-hand threaded screw forward in a counterclockwise direction.

Figure 7-9.
When you select the **Tapped Hole** radio button, the **Threads** area appears to provide thread options.

Pick the **Tapped Hole** radio button to reveal the **Threads** area of the **Hole** dialog box
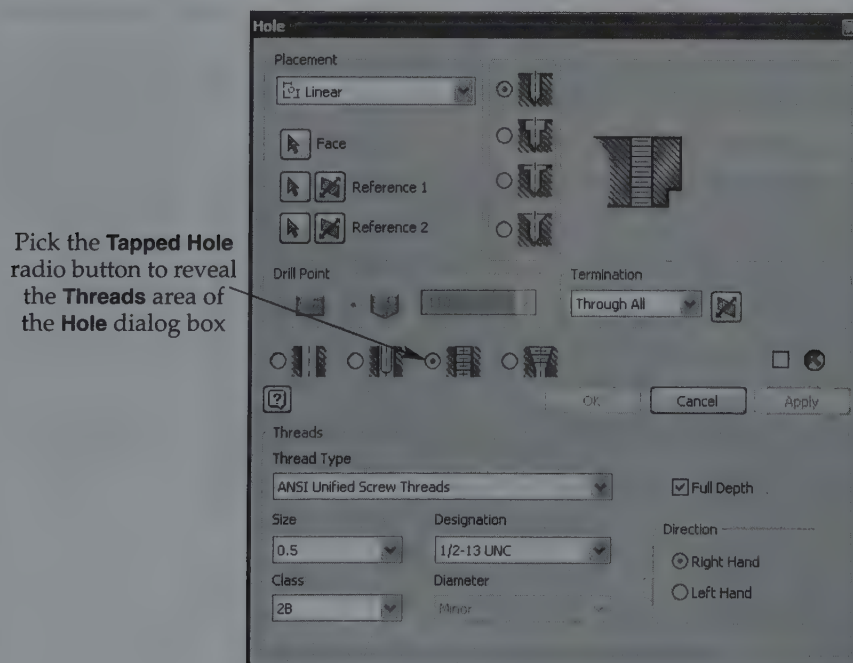
The **Size** drop-down list allows you to select the *nominal size*, which is usually the same as the major diameter of the threaded hole. Use the **Designation** drop-down list to choose the thread description, including *pitch*. For ANSI Unified Screw Threads, the specified number of threads per inch determines the pitch. For ANSI Metric M Profile threads, the specified pitch determines the number of threads.

> **NOTE**
>
> The **Hole** dialog box only provides designation options that correspond to the selected size.

The **Class** drop-down list allows you to specify the *thread class*. For ANSI Unified Screw Threads, 3, 4, 5, 6, 7, 8, or 9 may be available depending on the selected size and designation. For ANSI Metric M Profile threads, 1, 2, or 3 may be available depending on the selected size and designation. For metric threads, 1 is a coarse tolerance, 2 is a moderate tolerance, and 3 is a fine thread tolerance. For holes, the letter B follows the class to indicate internal threads.

Use the **Diameter** drop-down list or display box to indicate the diameter of the nominal size as **Minor**, **Pitch**, **Major**, or **Tap Drill**. The value references the setting in the **Tapped Hole Diameter** drop-down list in the **Modeling** tab of the **Document Settings** dialog box. Although the major diameter typically defines the nominal size of a hole, you must select the **Minor** option from the **Tapped Hole Diameter** drop-down list in order for threads to function correctly in a drawing.

Tapered Tapped Hole

Pick the **Tapered Tapped Hole** radio button to create *tapered threads* using options in the **Threads** area. Select a thread standard, such as **NPT** (National Pipe Thread), from the **Thread Type** drop-down list. Then choose the nominal hole size from the **Size** drop-down list. Use the **Direction** area to specify thread direction. The **Designation** drop-down list functions as a display box. All other options are unavailable because the selected standard and size control thread parameters.

> **NOTE**
>
> When you apply threads to a hole, a bitmap thread image appears on the hole walls. This representation is adequate for most applications. Use the **Coil** tool to create actual detailed threads.

Clearance Hole

Pick the **Clearance Hole** radio button to size the hole according to a specific fastener using options in the **Fastener** area. See **Figure 7-10**. Choose a standard, such as **ANSI Unified Screw Threads** or **ANSI Metric M Profile**, from the **Standard** drop-down list. Then use the **Fastener** drop-down list to pick a fastener based on the selected standard. The **Size** drop-down list contains multiple fastener size options based on the fastener. Use the **Fit** drop-down list to select a **Close**, **Normal**, or **Loose** fit, depending on the application. Pick the **OK** button to create the hole feature.
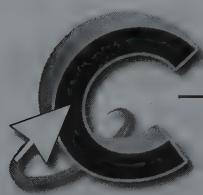
Exercise 7-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 7-1.

Figure 7-10.
When you select the **Clearance Hole** radio button, the **Fasteners** area appears to provide thread options associated with a fastener.

Pick the **Clearance Hole** radio button to reveal the fastener area

The hole diameter automatically changes when the fastener size or fit changes
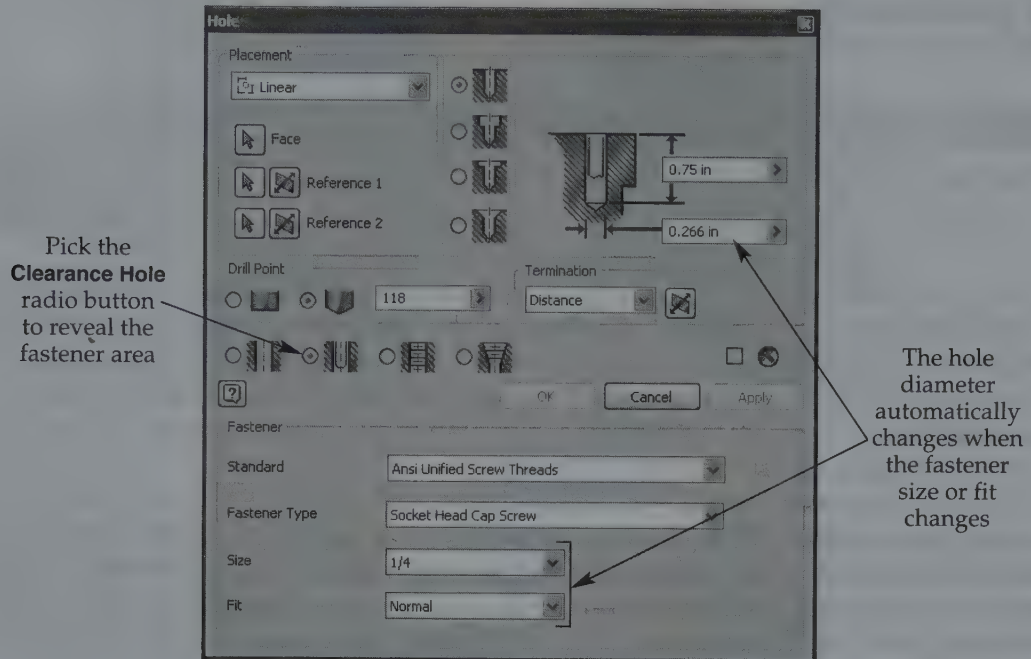
Bends

The **Bend Part** tool allows you to replicate the process of using a press or break to bend a part. See **Figure 7-11A**. You can also create a bend to add general curvature to a part, such as the curve applied to the handle shown in **Figure 7-11B**. To create a bend, you must first sketch a line to define where the bend occurs. Most often, the sketch is a single line, as shown in **Figure 7-11**, or a projected edge.

Access the **Bend Part** tool to create a bend using the **Bend Part** dialog box. See **Figure 7-12**. The **Bend Line** select button is initially active, allowing you to pick the sketched bend line. Then pick a bend option from the drop-down list based on known information. Specify values using the appropriate text boxes. **Figure 7-13** shows examples of each option.

By default, the bend occurs to the left of the bend line. Pick the **Bend right** button to bend the part to the right of the bend line. Another option is to pick the **Bend both** button to divide the bend equally on both sides of the bend line. Pick the **Flip bend** direction button if the part appears to be bending in the wrong direction. In some cases, you may need to pick the **More** button to access the **Bend Minimum** check box. Inventor checks the box by default, allowing only those features on the bend line to bend in situations when the bend line could project onto other features. Deselect the check box to bend all features. See **Figure 7-14**.

Pick the **OK** button to create a single bend. If you prepared additional separate sketches to create other bends, pick the **Apply** button and continue producing part bends as needed. Pick the **OK** button to exit the tool.

NOTE

A bend line sketch can be open and unconstrained, and it does not have to extend across the entire length of the part to be bent. However, use constraints to ensure design intent and to create an accurate bend.

Figure 7-11.
A—An example of a tool bent to replicate the process of using a press. B—An example of a handle bent to create curvature, in this case adding ergonomic characteristics.

Bend part sketch

.500

Initial Part

Bent Part

A

Bend part sketch

1.500

Initial Part

Bent Part

B
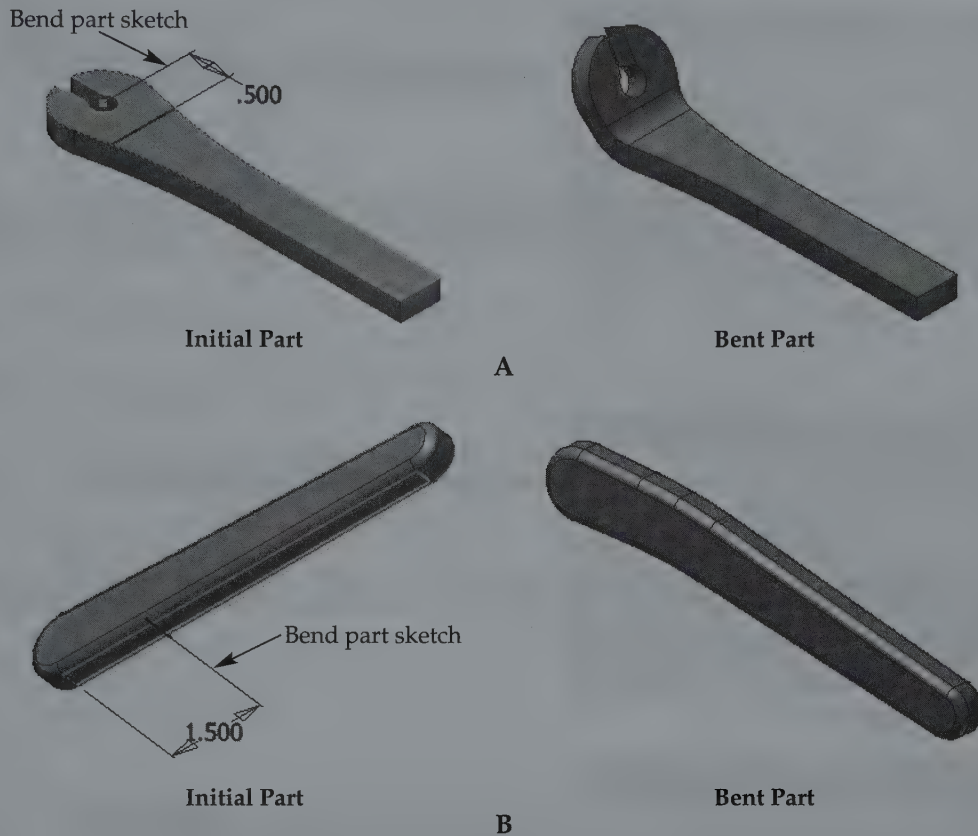
Figure 7-12.
Use the **Bend Part** dialog box to define the bend location and parameters.

Bend the part left
of the bend line

Bend the part right
of the bend line

Bend both sides
of the part

Choose a set
of bend
specifications
from the list

Pick to flip the
bend above or
below the bend line
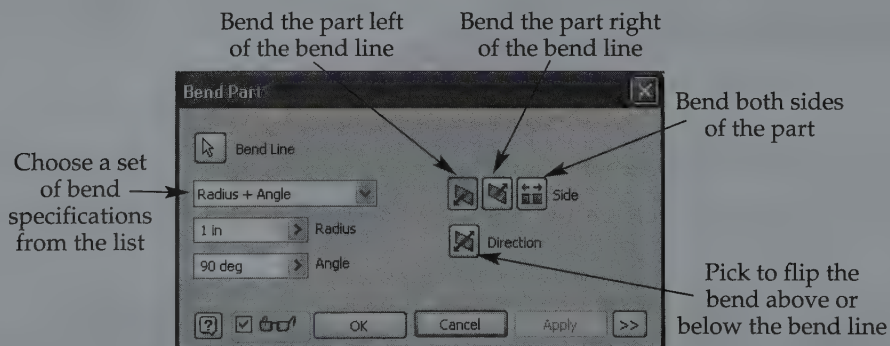
Figure 7-13.
Examples of selecting and entering bend specifications. Choose the bend option based on the information you know.
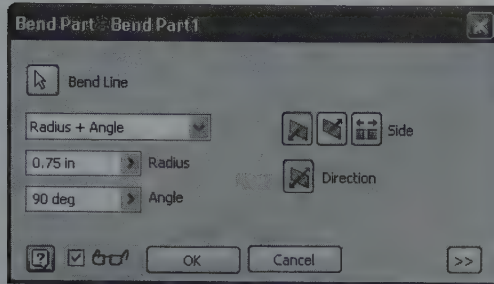
Radius + Angle
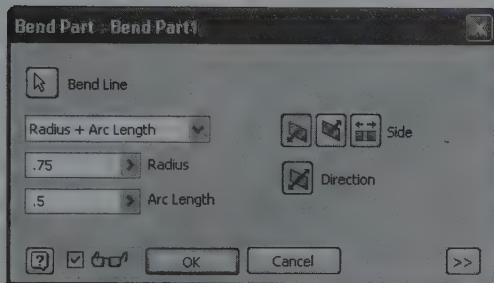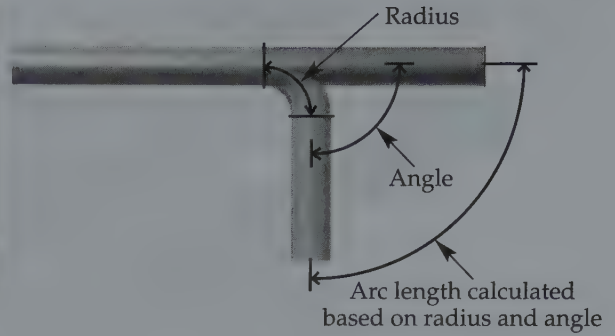
Radius + Arc Length
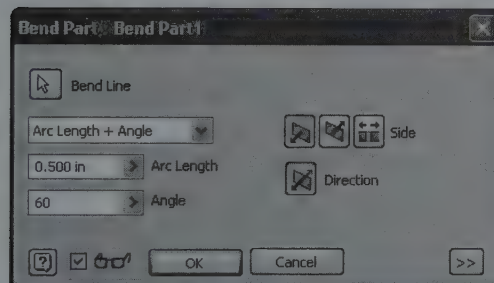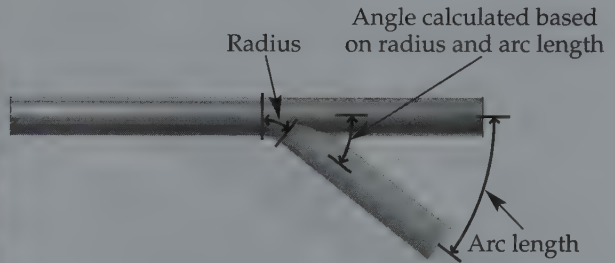
Arc Length + Angle
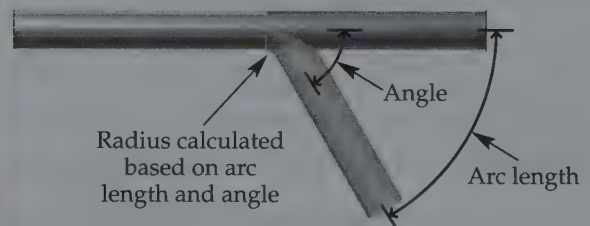
Figure 7-14.
Use the **Bend Minimum** check box to control how the bend reacts when other features could be bent by the bend line projection.

Bend Minimum Checked

Bend Minimum Unchecked
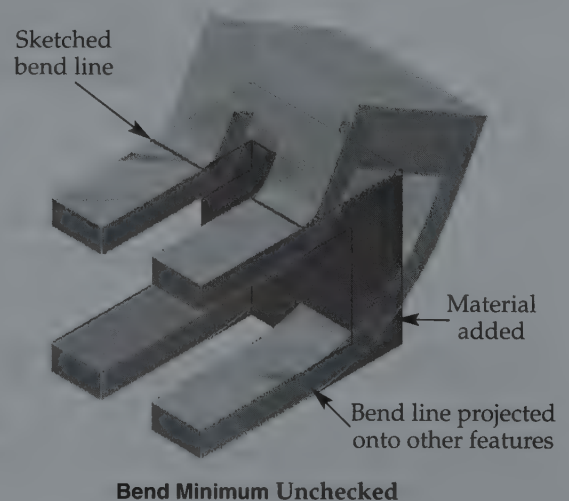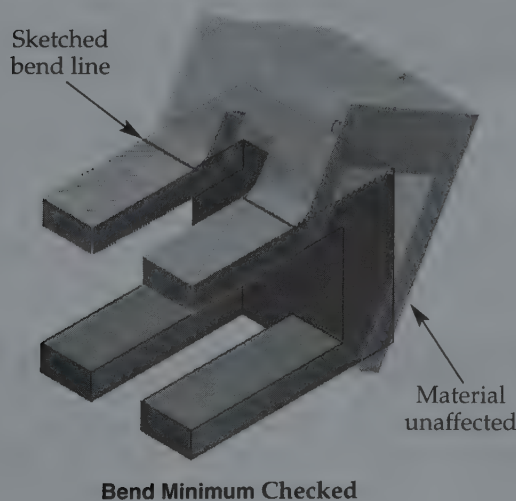
Exercise 7-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 7-2.

Ribs and Webs

Ribs and *webs* are similar features that you can create from the same sketch geometry. The ribs and webs shown in **Figure 7-15** constitute a *network* of ribs or webs. A rib or web sketch is most often an open loop and may only include a single line or arc. The feature defines all other parameters. You can form a rib or web from an unconstrained sketch, as shown in **Figure 7-16A**, but as with other sketches, you should add constraints for definition and stability, as shown in **Figure 7-16B**. When creating a rib or web network, sketch all profiles on the same sketch plane.

rib: A closed section of material usually added to reinforce a part without adding excessive material or weight.

web: An open section of material usually added to reinforce a part without adding excessive material or weight.

network: Several ribs or webs created using the same direction and thickness.

PROFESSIONAL TIP

To avoid complications, only the lines that will form ribs and webs should use the sketch geometry format. Convert all other sketch geometry to the construction format.

Figure 7-15.
An example of ribs and webs created from the same sketch.

3.000

3.000

Change all projected or reference geometry to construction format
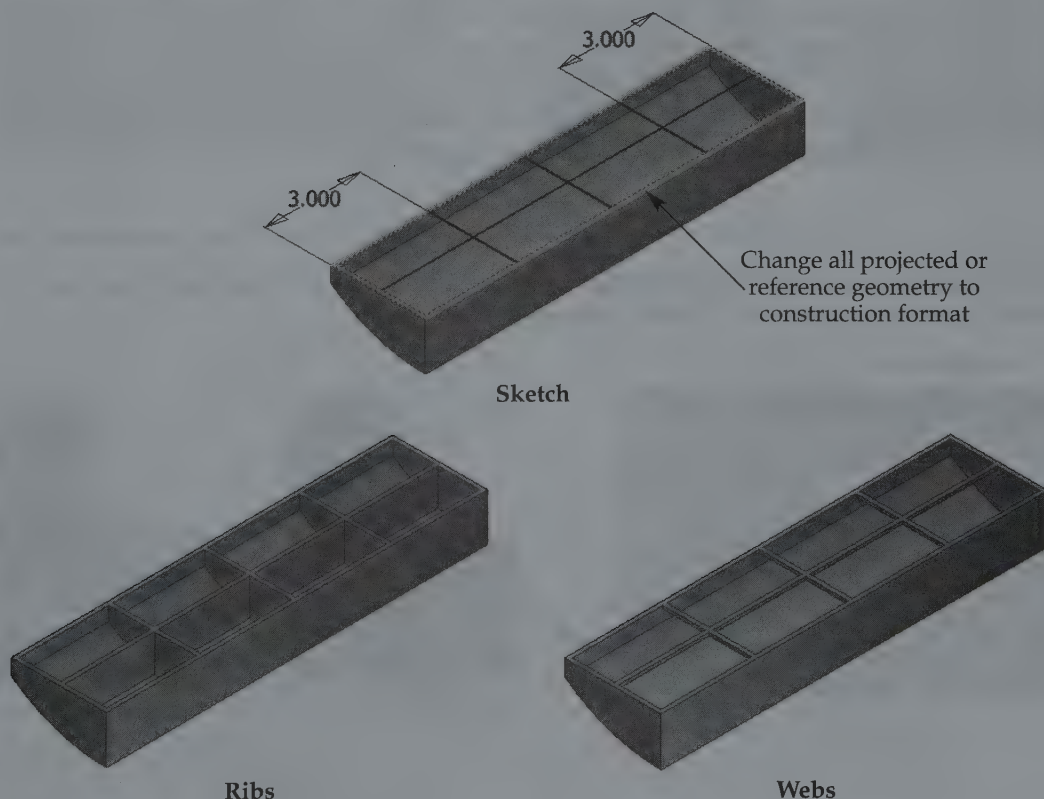
Sketch

Ribs

Webs

Figure 7-16.
A—An example of an open-loop, unconstrained sketch used to create ribs and webs. B—An example of a constrained sketch, referencing existing geometry, used to create ribs and webs. The sketch plane is visible for reference.

.200

.150

A   B

Convert to
construction
geometry
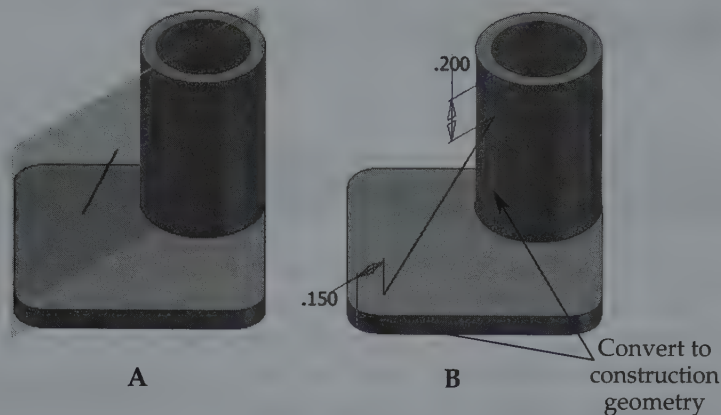
NOTE

Create a rib or web network using a single sketch, as shown in **Figure 7-15**, or using a feature pattern.

Access the **Rib** tool to create a rib or web using the **Rib** dialog box. See **Figure 7-17**. If a sketch includes one profile, the profile is selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile(s) to rib or web. Next, use the **Direction** button to specify where the rib or web occurs. Direction arrows and a preview appear when you move the cursor. Multiple solutions are often possible. When it appears that the arrows are pointing in the correct direction and the preview looks acceptable, pick to select. Use the **Thickness** text box to enter or select the rib or web width. Then you may have to pick one of the side buttons to define the side of the profile on which the feature occurs. If necessary, use the **Taper** text box to add taper to the feature beginning at the sketch plane.

Figure 7-17.
The **Rib** dialog box with the **To Next** extents button selected. The rib direction arrow and preview appear when you pick a sketch and select a direction.

To Next Button

Rib

Shape	Thickness	Extents
Profile	0.1 in	
Solid		
Direction		Taper

☑ Extend Profile

OK   Cancel

.200

.150

Preview   Rib
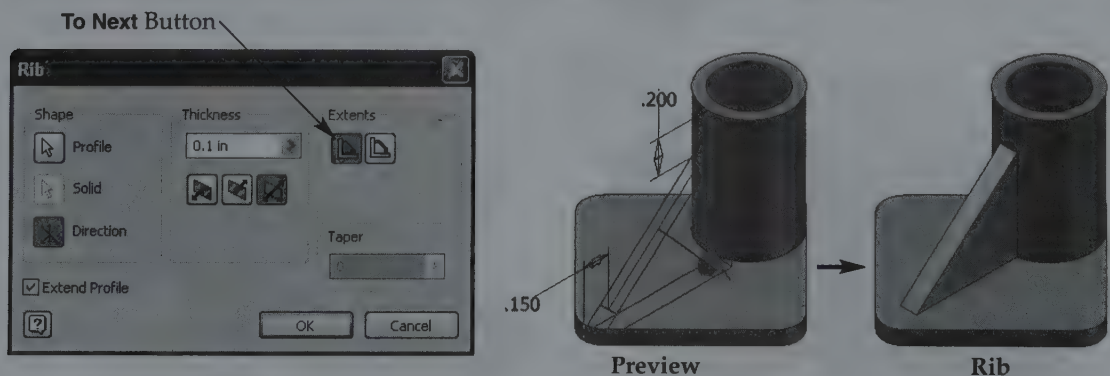
By default, the **To Next** extents button is active to create a rib. To create a web, pick the **Finite** extents button to display the **Extents Depth** text box, and specify the web thickness, or depth. See **Figure 7-18**. The **Extend Profile** check box is most often associated with web creation. Check the box to extend the ends of the web to intersecting features. If this box is not selected, the web terminates at the ends of the sketch profile. Pick the **OK** button to create the rib or web.

Exercise 7-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 7-3.

Figure 7-18.
Use the **Finite** extents button to create a web.
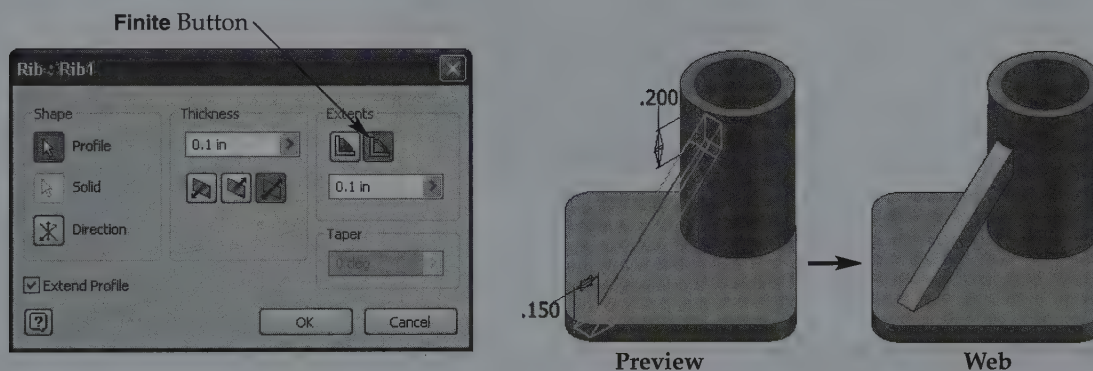
Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. List three ways to locate a hole.
2. Identify an effective method for placing a hole concentric to a cylindrical or conical feature without using a separate sketch.
3. Briefly describe a drilled hole.
4. Describe a counterbored hole.
5. What is a spotface?
6. Describe a countersunk hole.
7. Explain the difference between a right-hand and left-hand thread.
8. Explain the meaning of the term *nominal size*.
9. Define *pitch*.
10. What is the thread class?
11. Describe a common use for part bends.
12. In which direction does a bend occur by default?
13. What should you do if you are creating a bend that might interfere with existing features to blend the bend with the existing features?
14. What is the difference between ribs and webs?
15. Briefly describe a network of ribs and webs.

Problems

1–4 Instructions:

- *Open the specified part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **File:** P6-3.ipt
   Save as: P7-1.ipt
   Title: SWIVEL
   Specific instructions: Use the **Concentric** placement option of the **Hole** tool to apply the counterbored hole shown. Place it on the bottom face of the extrusion. Use the hole note given to define the size of the hole.

Ø6.35 THRU
⌴ Ø9.50 ▼ 3.20
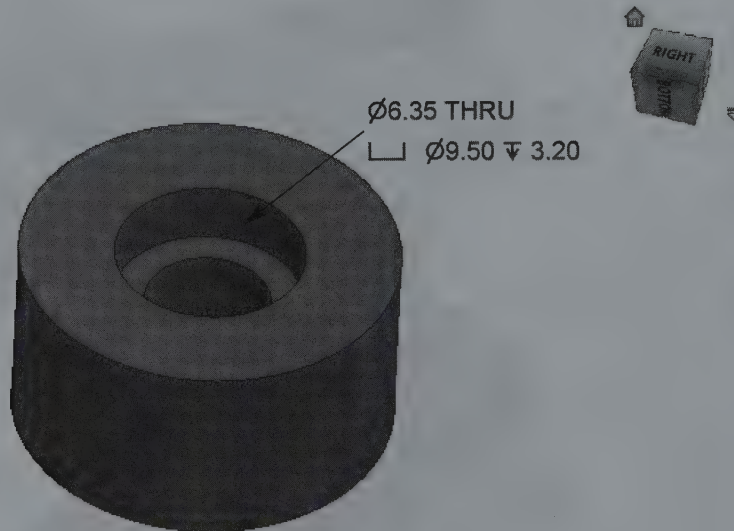
2. **File:** P6-6.ipt
   Save as: P7-2.ipt
   Title: BODY
   Specific instructions: Use the **Linear** placement option of the **Hole** tool to apply the drilled and threaded hole shown using the dimensions provided. Use the **To** termination option to cut the hole completely through to the top portion of the C-clamp body.

9.5

9.5

M10 x 1.5 – 6H
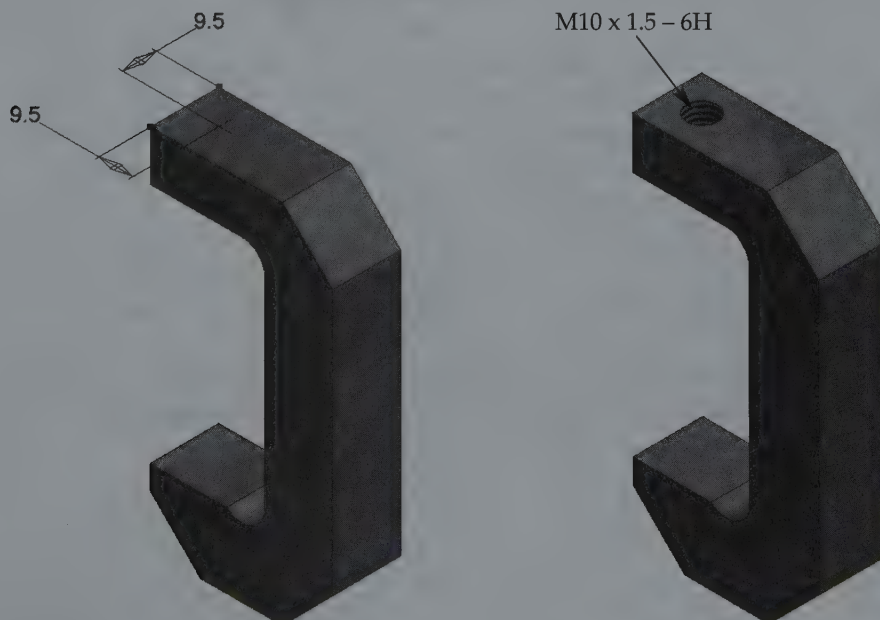
3. **File:** P6-9.ipt
   Save as: P7-3.ipt
   Title: STOP
   Specific instructions: Open a sketch on the face shown and sketch the **Point, Center Point** according to the information given. Access the **Hole** tool and use the **From Sketch** placement option to position a hole at the sketched **Point, Center Point**. The hole is a simple hole with a diameter of .0864, cut through the entire part using the **Through All** termination option.

Point coincident to projected edge

Projected edge; convert to construction line

Projected edge; convert to construction line

.085
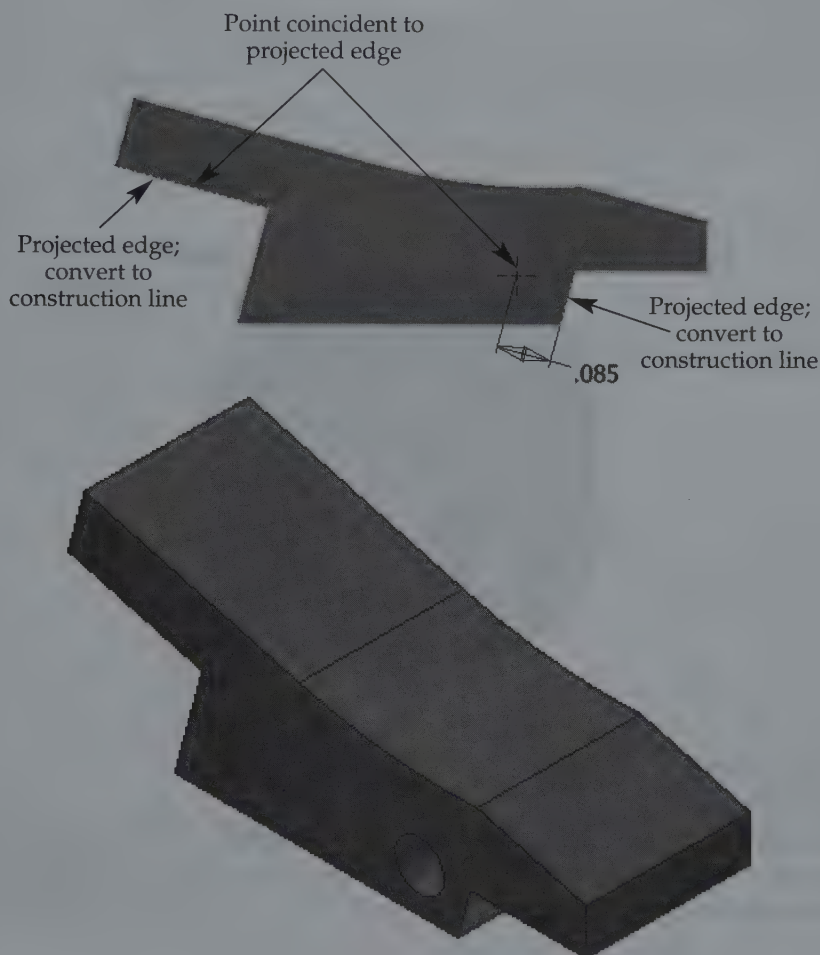
4. **File:** P6-5.ipt
   Save as: P7-4.ipt
   Title: SUPPORT BRACKET
   Specific instructions: Use the **Concentric** placement option of the **Hole** tool to apply the simple counterbored hole on the face of the extrusion as shown. Use the hole note to define the size of the hole. Open a sketch on the face of the lower small, extruded cylinder and project the faces as shown. Change the projected edges to construction lines. Use the **From Sketch** placement option of the **Hole** tool to apply the threaded countersunk holes at the centers of the projected cylinder faces as shown. You must pick the centers. Use the hole note and the **Through All** termination option to define the size of the hole.

Project these faces onto the sketch plane

⌀0.125 ▽ 0.250
⌴ ⌀0.250 ▽ 0.125

0-80 UNF - 2B
∨ ⌀0.125 X 60°

Sketch on this face

5–7 Instructions:
- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects at the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

5. **Title:** BENT PULLER
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-010-01
   Project: BENT PULLER
   Material: Stainless Steel
   Color: As Material
   Save as: P7-5.ipt
   Specific instructions: Refer to the illustrations on the following pages. Open a sketch on the XZ plane and create the sketch geometry shown. Extrude the sketch .25 in the positive direction. Add the sketch shown on the top face, and then cut-extrude the sketch through all. Add the bend line sketch on the top face as shown. Use the **Bend Part** tool to bend the part at the sketched line according to the following specifications and the model shown: .25 radius, 60° angle, bend left and up.

Projected center point, coincident to line midpoint

.500

Vertical

Parallel

4.000

7.500

Tangent

Tangent

.500

.300

This value is half of the .200 parameter

fx:.100

.200

Bend line

.500

Projected edges
converted to
construction lines

TOP
FRONT RIGHT
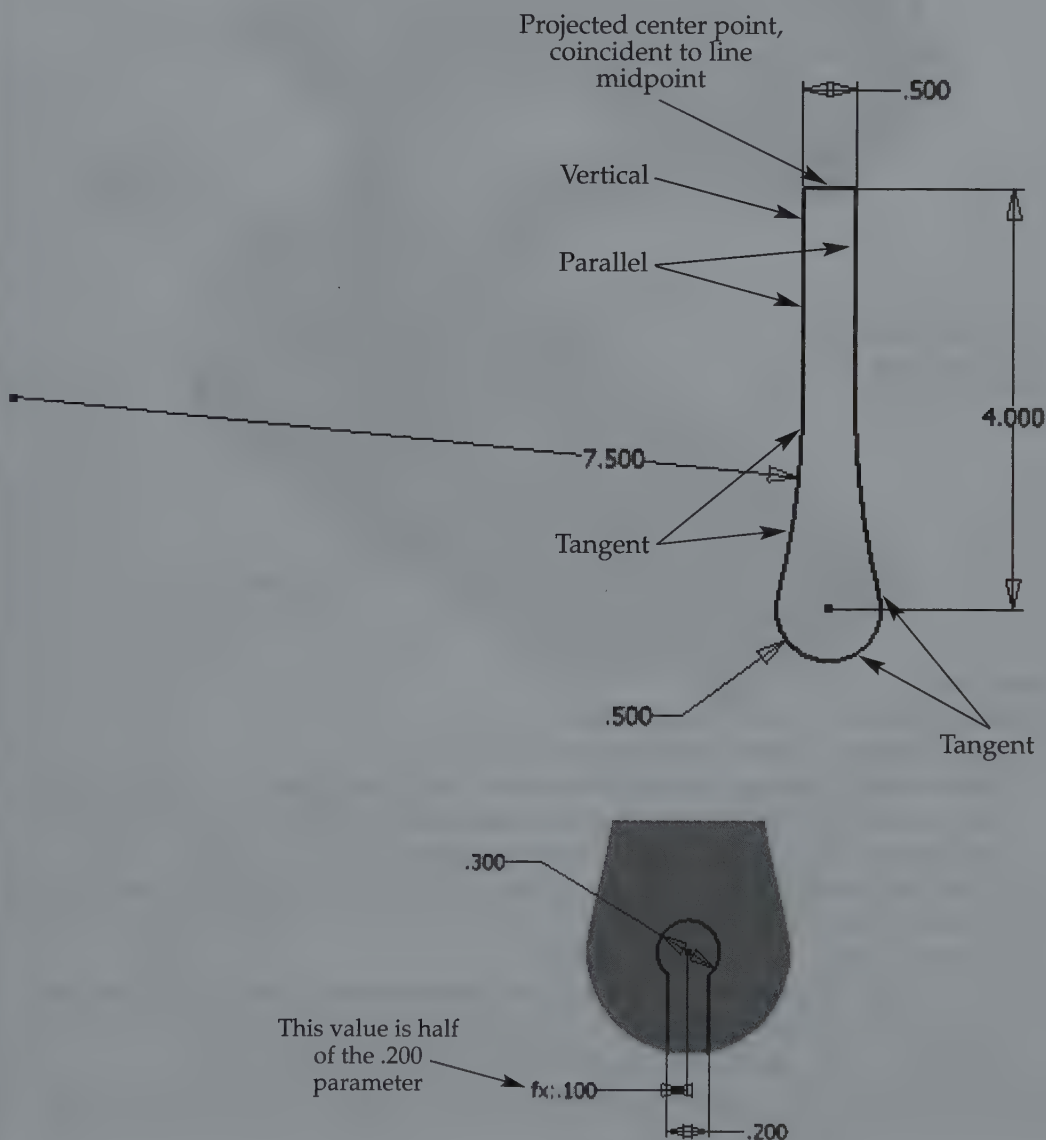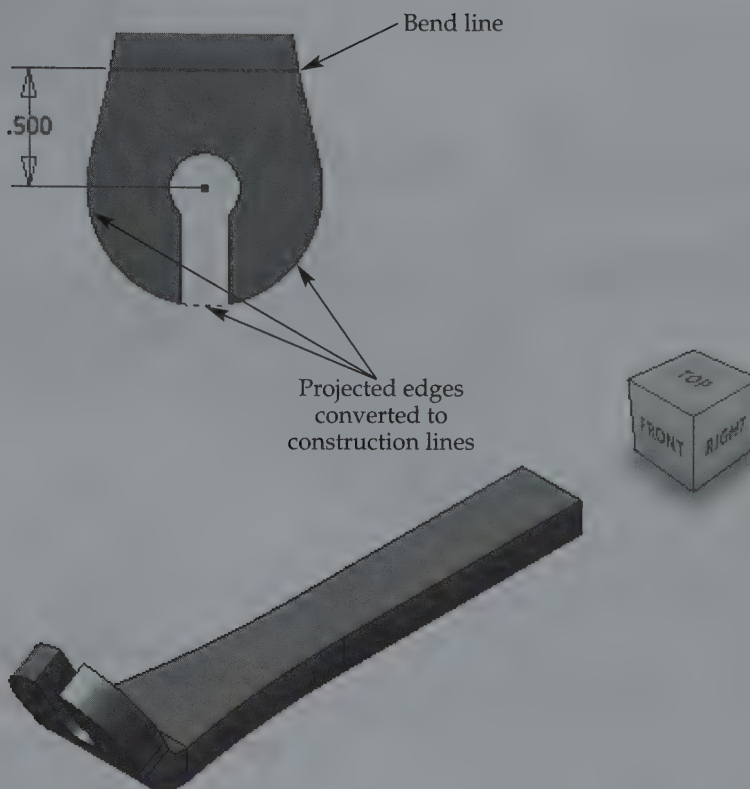
6. **Title:** CHISEL HANDLE
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-011-01
   Project: CHISEL
   Material: Choose a material
   Color: Choose a color
   Save as: P7-6.ipt
   Specific instructions: Use dimensions of your choice to design and build the original chisel handle shown in Figure 7-11B, without the rounded edges. Use the bend tool to add curvature to the part as shown in Figure 7-11B.

7. **Title:** BRACKET
   Units: Metric
   Template: Part-mm.ipt
   Part Number: IAA-012-01
   Project: BRACKET
   Material: Aluminum-6061
   Color: Blue Sea
   Save as: P7-7.ipt
   Specific instructions: Create the part shown using the specified features. When complete, your part should contain at most three extrusions from three sketches, three ribs from three sketches, two webs from one sketch, and two holes from one sketch.
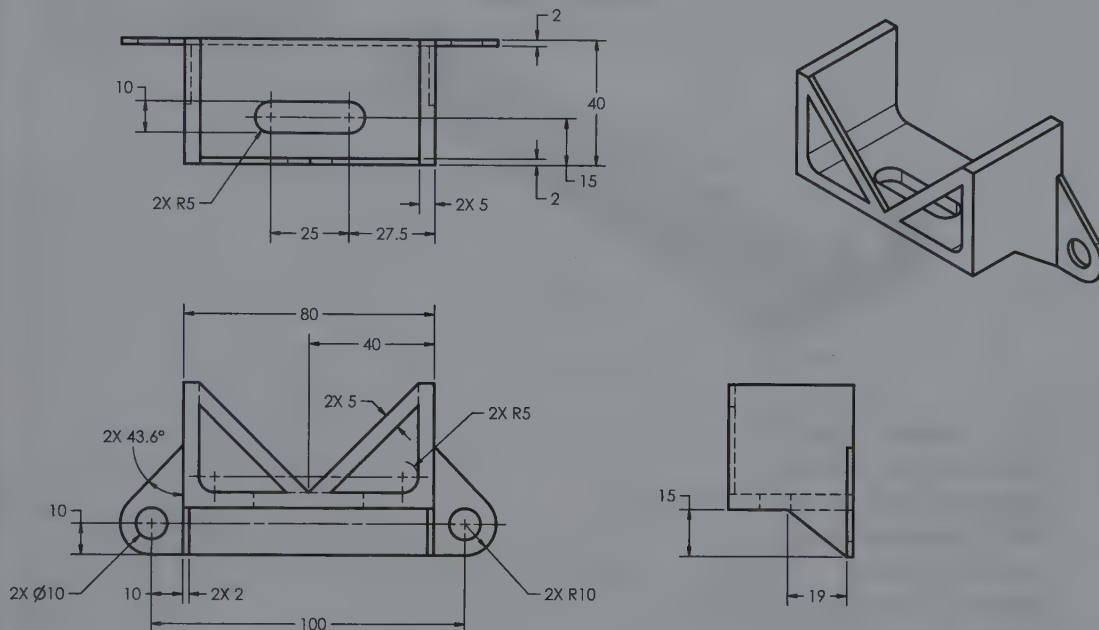
Fillets, Rounds, and Chamfers

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Place fillet and round features.
✓ Add chamfer features.

You can generally add a ***placed feature*** to a model if an appropriate base feature is present. Placed features often help finalize a design or add to sketched feature geometry. This chapter describes three common placed features: fillets, rounds, and chamfers.

placed features: Features added to an existing feature without using a sketch.

Edge Fillets and Rounds

Fillets often ease or simulate the machining of inside corners to help release patterns from castings and forgings, or to relieve stress. *Rounds* typically remove sharp machined edges and help release patterns from forgings and castings. **Figure 8-1** shows an example of a part with multiple fillets and rounds. You can place fillets and rounds on selected edges or at the intersection of selected faces.

fillet: A curve placed at the inside intersection of two or more faces, adding material to a feature.

round: A curve placed on the exterior intersection of two or more faces, removing material from a feature.

Figure 8-1.
The part feature **Fillet** tool allows you to place fillet and round features.
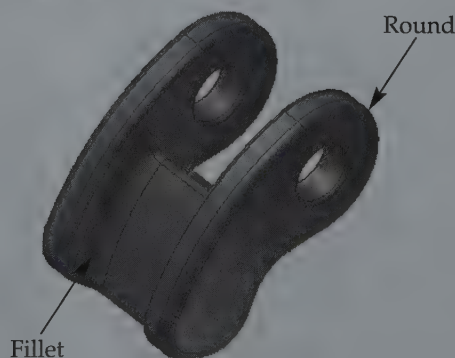
Round

Fillet

constant fillets and rounds: Fillets and rounds that have a curve radius that does not change.

variable fillets and rounds: Fillets and rounds that have different curved radii placed at precise points between the start and end of a feature edge.

Access the **Fillet** tool to create fillets and rounds using the **Fillet** dialog box. See **Figure 8-2**. Pick a button in the upper-left corner of the **Fillet** dialog box to display options for placing edge, face, or full radius fillets and rounds. The **Edge Fillet** button is active by default and allows you to create *constant* and *variable* fillets and rounds on selected edges. See **Figure 8-3**. Use the **Edge Fillet** option to create different types of fillets or rounds without continually re-accessing the **Fillet** tool. You can add all of the fillets and rounds shown in **Figure 8-3**, for example, using a single fillet operation, which displays as a single feature in the browser.

> **PROFESSIONAL TIP**
>
> If you plan to suppress certain fillets or want to have additional fillet features, access the **Fillet** tool more than once.

Figure 8-2.
Use the default **Edge Fillet** button of the **Fillet** dialog box to add constant and variable fillets to selected edges.

Pick to create an edge fillet

Pick to create a face fillet

Pick to create a full round fillet
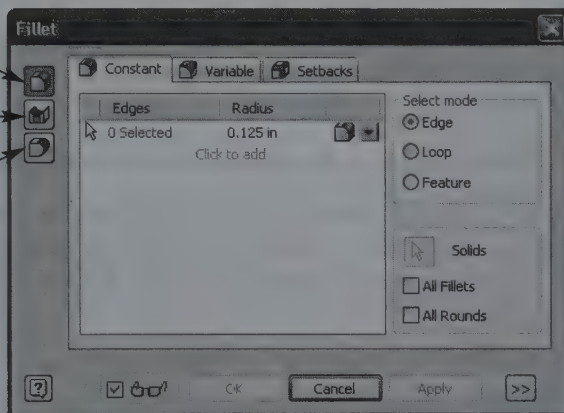
Figure 8-3.
An example of a part with constant and variable fillets and rounds. By accessing the **Fillet** tool once, you can apply several fillet and round styles, all having different radii.

Variable fillet   Constant round

Variable round

Constant fillet
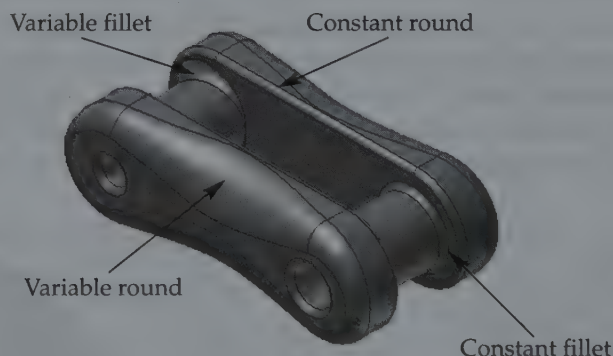
Fillet and Round Options

Before you place fillets and rounds, you should understand how the options in the **More** area of the **Fillet** dialog box affect fillet and round creation. The **Roll along sharp edges** check box, which is unselected by default, maintains the specified radius by forming a lip when you fillet an object around a sharp edge. To keep the original edge geometry, but adjust the radius, select the **Roll along sharp edges** check box. See **Figure 8-4**.

The **Rolling ball where possible** check box, which is selected by default, simulates the process of using a convex milling cutter (for fillets) and corner-rounding milling cutter (for rounds). The result of selecting or deselecting the **Rolling ball where possible** check box is most evident when you fillet or round three intersecting edges. See **Figure 8-5**. The **Automatic Edge Chain** option allows you to select a chain of tangent edges with a single pick. Deselect the **Automatic Edge Chain** check box to pick individual segments. See **Figure 8-6**. Select the **Preserve all features** check box to ensure that Inventor accounts for the features that intersect the fillet and to determine intersection redefinition information when creating fillets and rounds.

> **NOTE**
>
> Most fillet and round options apply to edge fillet development. Only the **Preserve all features** check box is selectable for creating face and full round fillets.

Figure 8-4.
Notice the change to the edge when you select or deselect the **Roll along sharp edges** check box.

Edge rises to form specified radius

Radius adjusts to edge geometry

☐ Roll along sharp edges   ☑ Roll along sharp edges
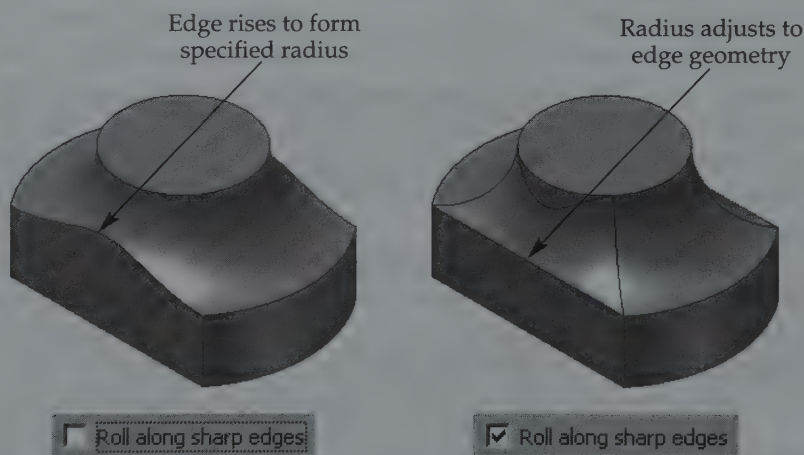
Figure 8-5.
Compare the effects when the **Rolling ball where possible** option is and is not active.

☑ Rolling ball where possible   ☐ Rolling ball where possible
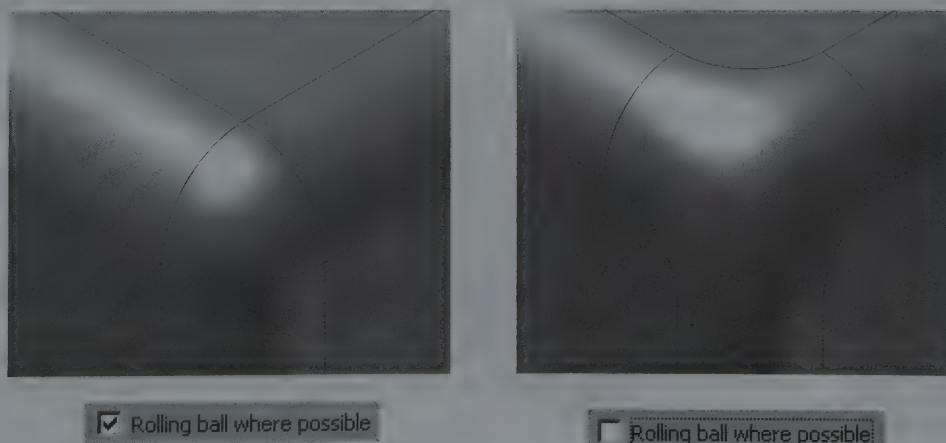
Figure 8-6.
You can either select individual edges or select with an edge chain to pick all tangent edges.

Three distinct edges were selected with an edge chain

Two individual edges were selected separately

☑ Automatic Edge Chain

☐ Automatic Edge Chain
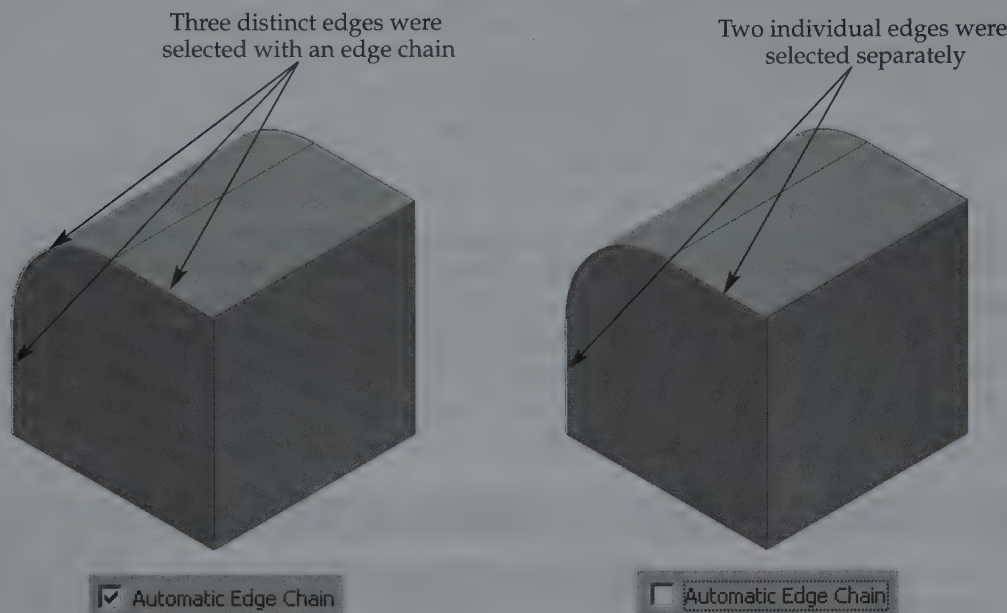
Constant Fillets and Rounds

Use the options available in the **Constant** tab, shown in **Figure 8-2**, to apply constant fillets and rounds. First, specify a radius using the **Radius** column text box. Then pick the **Tangent Fillet** or **Smooth (G2) Fillet** button from the flyout to produce the desired fillet or round transition. A tangent fillet is acceptable for most applications and produces a curve tangent to adjacent faces. Select the **Smooth (G2) Fillet** option to create a G2 curve, or curvature-continuous situation, between the curve and adjacent faces.

Next, pick the appropriate **Select Mode** radio button to help select single edges, a closed loop, or an entire feature, and then pick the object to fillet or round. A feature preview appears by default. To select and fillet all interior edges using the same style and radius, pick the **All Fillets** check box. To select and round all exterior edges using the same style and radius, pick the **All Rounds** check box.

> **NOTE**
>
> To deselect objects, hold down the [Ctrl] key and pick the items to remove from the selection set. To remove all selections associated with a specific radius, pick the edges to remove in the **Edges** column and press [Delete].

To apply fillets and rounds with different radii using a single **Fillet** operation, pick the **Click to add** button. Then enter a different radius and select edges. Continue adding fillets and rounds as required. Pick the **Apply** button to create the feature and remain in the **Fillet** tool, or pick the **OK** button to create the feature and exit.

> **NOTE**
>
> You can add variable fillets and rounds and adjust setbacks in addition to constant fillets and rounds using the options in the **Variable** and **Setbacks** tabs.
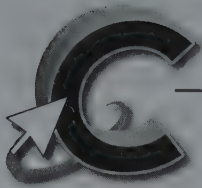
Exercise 8-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 8-1.

Variable Fillets and Rounds

Use the options available on the **Variable** tab, shown in **Figure 8-7**, to apply variable fillets and rounds. First, select an edge. The start and ends are automatically selected unless you pick a closed loop, such as a cylinder edge, in which case you must pick points. Next, pick additional points along the selected edge. A yellow dot and preview identify the location of each new point.

Once you select all points, pick a point in the **Point** column. A blue dot identifies the selected point in the model for reference. Specify the radius of the point using the **Radius** column text box. Precisely define the location of the point using the **Position** column text box. The position of additional points is relative to the start point and the individual edge pieces, not a tangent loop. Remember, the active point displays a blue dot on the selected edge. Adjust the radius and position of each point. If necessary, select additional points along the edge and adjust the radius and position of each.

Figure 8-7.
Using the **Variable** tab of the **Fillet** dialog box to create variable fillets and rounds. Notice the point style for selecting points for a variable-radius fillet or round.
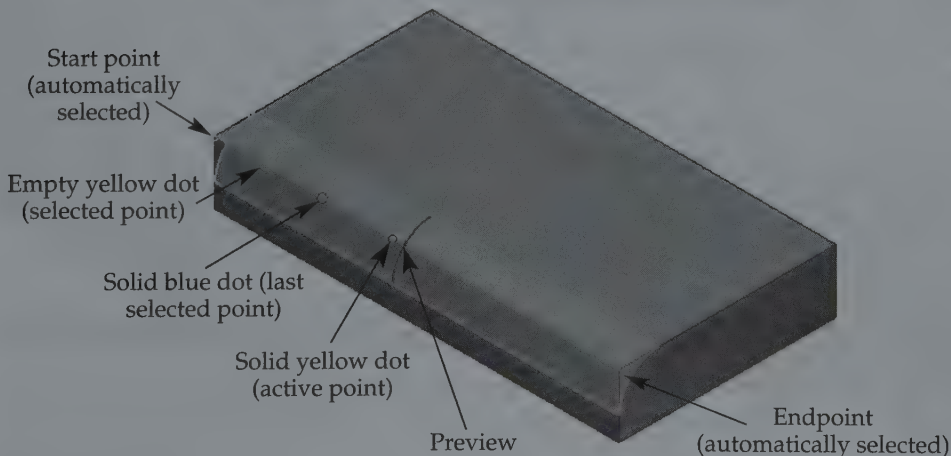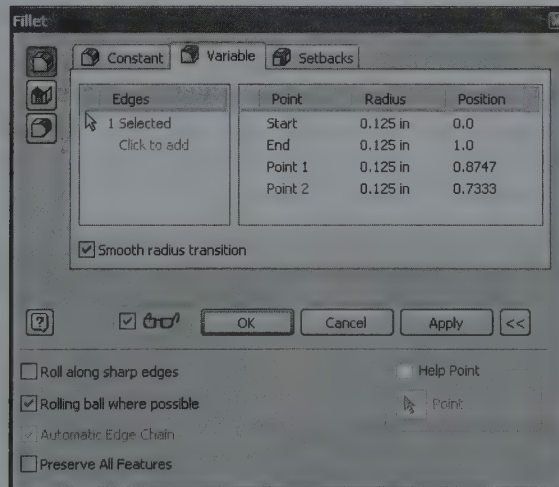
The **Smooth radius transition** check box is selected by default and creates a steadily increasing or decreasing blend between variable points. To form a direct, linear arc between variable points, deselect the **Smooth radius transition** check box. To apply additional variable fillets and rounds on other edges, pick **Click here to add** and repeat the process of selecting and defining points. Pick the **Apply** button to create the feature and remain in the **Fillet** tool, or pick the **OK** button to create the feature and exit.

NOTE

To delete a variable fillet or round edge or point listed in the **Edges** or **Point** columns, pick the edge or point to remove and press [Delete].

Controlling Setbacks

setback: Point at which a fillet or round on one edge begins to combine with a fillet or round of at least two other edges.

vertex: When referring to fillet and round setbacks, the intersection of three or more edges.

You can observe the process of modifying *setback* values when you select or deselect the **Rolling ball where possible** check box. Setbacks greater than the fillet or round radius blend curves beginning further from the *vertex*. Setbacks less than or equal to the fillet or round radius create a rolling ball effect. Refer again to **Figure 8-5**. Use the options available on the **Setbacks** tab, shown in **Figure 8-8**, to modify fillet and round setbacks.

Pick the corner, or vertex, of the feature edges to add the vertex to the **Vertex** column. A blue dot identifies the selected vertex in the model for reference. Each edge intersecting the selected vertex appears in the **Edge** column. If you pick one of the edge names, the matching edge in the model highlights. If you pick one of the edges on the feature, a cursor appears to the left of the corresponding edge name in the **Edge** column. Use the **Setback** column text box to specify the setback for each edge.

An alternative to entering setback values is to pick the **Minimal** check box in the **Minimal** column. This option, which applies the least amount of allowable setback at the selected vertex, is most noticeable when applied to edges with different radii. See **Figure 8-9**. To modify additional vertex setbacks, pick the **Click here to add** button and repeat the process of selecting a vertex and defining setbacks. Pick the **Apply** button to create the feature and remain in the **Fillet** tool, or pick the **OK** button to create the feature and exit.

Figure 8-8.
Using the **Setbacks** tab of the **Fillet** dialog box to control fillet and round setback. Pick a vertex to specify the edges to modify setback.
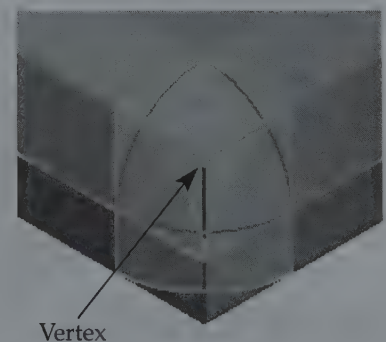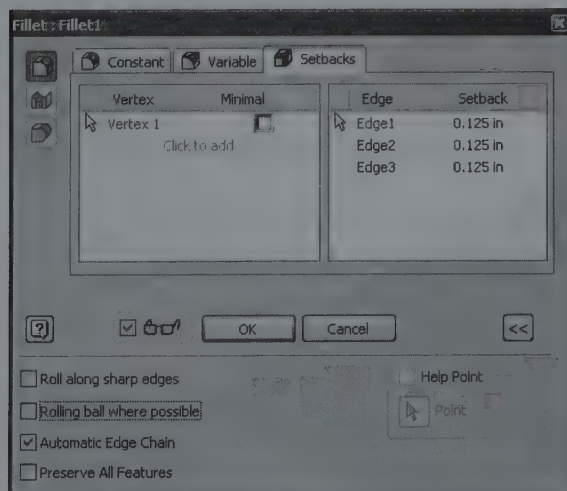
Figure 8-9.
A—An example of the default setbacks applied to intersecting rounds. B—Applying the **Minimal** option to blend rounds using the least amount of setback.

Default setbacks
applied when rounding
multi-radii edges

Minimal
setbacks

A

B
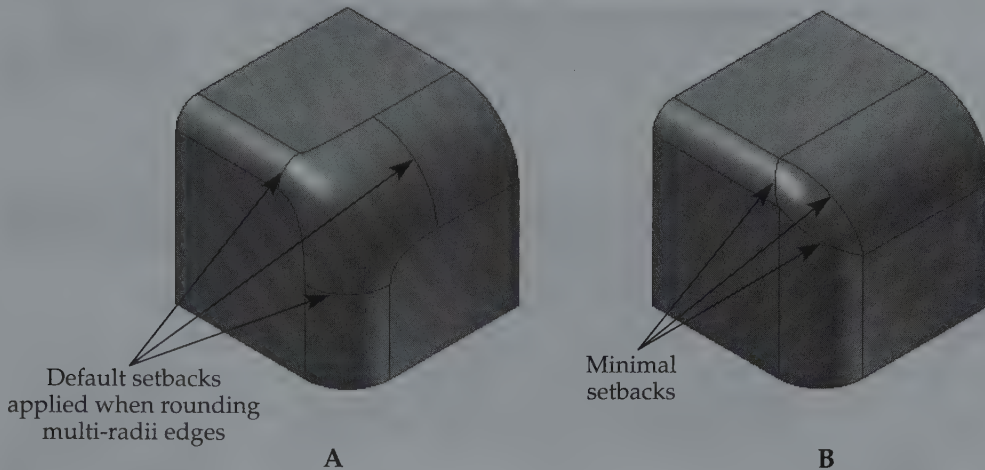
NOTE

To delete a setback vertex or edge listed in the **Vertex** or **Edge** columns, pick the vertex or edge to remove and press [Delete].

Exercise 8-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 8-2.

Face Fillets and Rounds

Access the **Fillet** tool and pick the **Face Fillet** button in the **Fillet** dialog box to add a fillet or round tangent to selected faces. See **Figure 8-10**. The **Face Set 1** button is active by default, allowing you to choose the first face tangent to the fillet or round. Next, with the **Optimize for Single Selection** check box selected, the **Face Set 2** button becomes activated, allowing you to choose the second face. Enter a radius in the **Radius** text box.

A preview appears if you enter an acceptable radius. If the preview does not look correct, you may be able to adjust the fillet or round by selecting the **Help Point** check box in the **More** area of the **Fillet** dialog box and then picking a location on the feature to add the help point. A help point can sometimes help you define the transition between the curve and feature faces. If the preview looks correct, pick the **Apply** button to create the feature and remain in the **Fillet** tool, or pick the **OK** button to create the feature and exit. **Figure 8-11** shows how the **Face Fillet** option can apply to fillets and rounds.

NOTE

If necessary, use the face set **Flip** buttons to flip the direction of the fillet or round when adding a face fillet or round between surfaces.

Figure 8-10.
Using the **Face Fillet** option of the **Fillet** dialog box to form rounds. Notice that the first face selected appears in a contrasting color from the second face selected.

Deselect to remove tangent adjacent faces from the selection set

Deselect to select a face set that includes multiple faces

First face selected

Second face selected

Original Feature (Revolution)   **Faces Selected**   **Resulting Round**
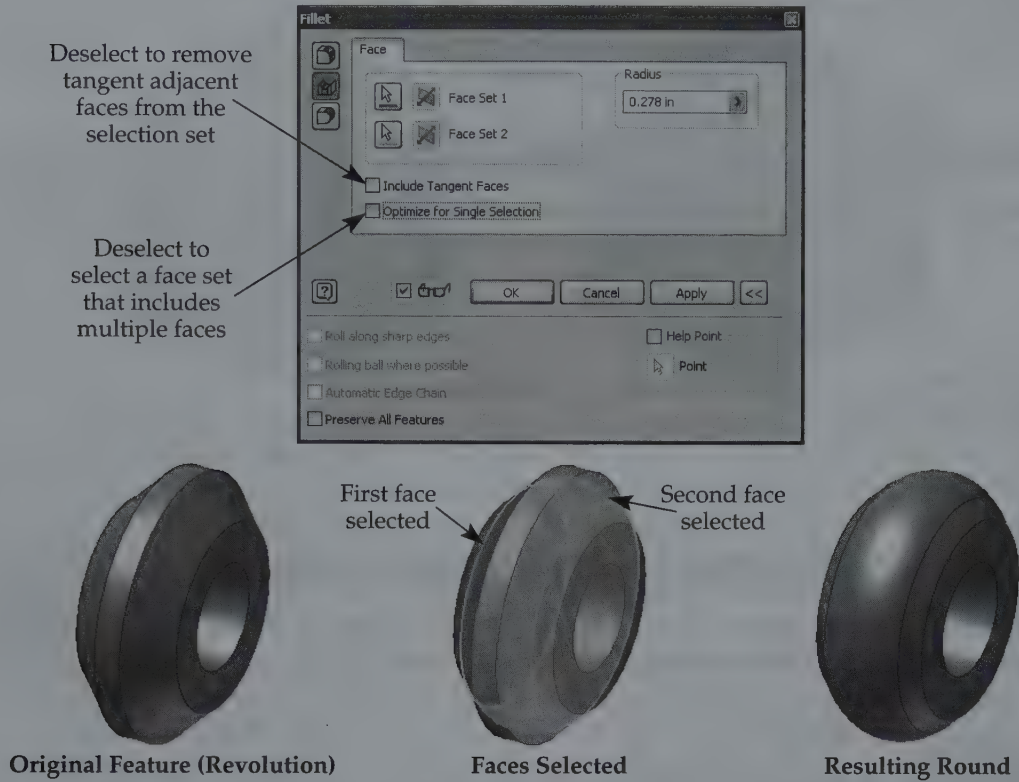
Figure 8-11.
Additional examples of fillets and rounds produced using the **Face Fillet** option.

Selected faces

Fillet

Forming a Fillet between Two Nonintersecting Feature Faces

Selected faces

Round

Forming a Round between Two Intersecting Feature Faces
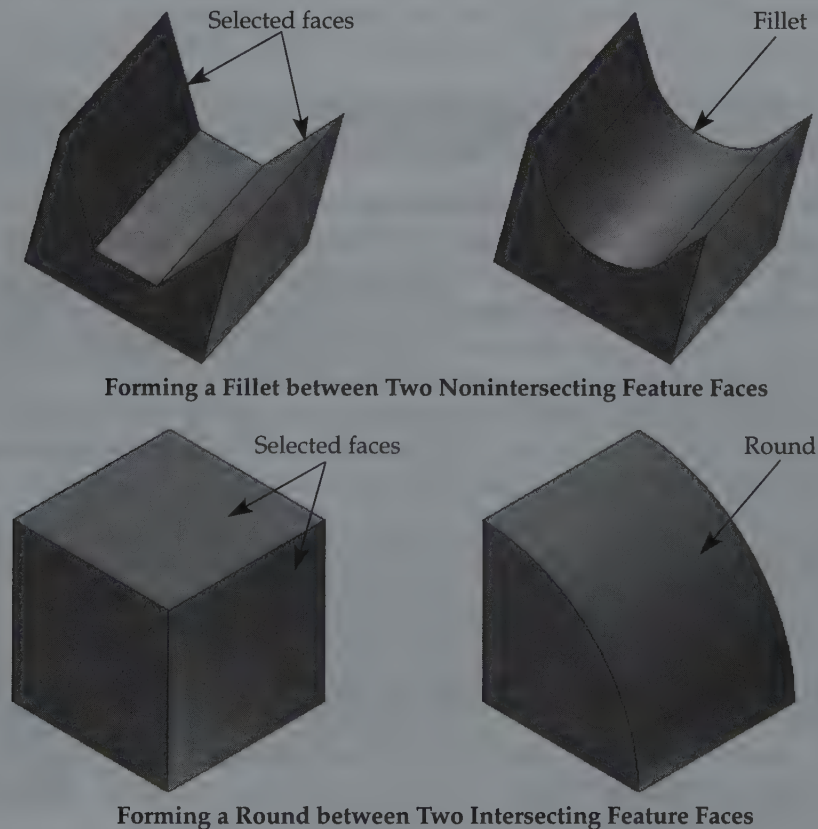
Exercise 8-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 8-3.

Full Radius Fillets and Rounds

Access the **Fillet** tool and pick the **Full Round Fillet** button in the **Fillet** dialog box to create a *full radius* fillet or round. See **Figure 8-12**. The **Side Face Set 1** button is active by default, allowing you to choose the first side face used to define the fillet or round. Next, with the **Optimize for Single Selection** check box selected, the **Center Face Set** button becomes activated, allowing you to choose the center face. Then the **Side Face Set 2** button is activated, allowing you to choose the second side face.

full radius fillets and rounds: Fillets and rounds controlled by the linear dimension of a feature, such as the thickness of a part or width of a slot, producing half of a circle or cylinder; most often associated with a round.

Figure 8-12.
Using the **Full Round Fillet** option of the **Fillet** dialog box to create a true full round.

Deselect to remove tangent adjacent faces from the selection set

Deselect to select a face set that includes multiple faces

Original feature (extrusion)

The first side face (first selection) appears in a contrasting color from the center and second face selections

Center face (second selection)

The second side face (third selection) is the bottom face of this part

Resulting round

A preview appears if you select appropriate faces. If the preview looks correct, pick the **Apply** button to create the feature and remain in the **Fillet** tool, or pick the **OK** button to create the feature and exit. **Figure 8-13** shows a hinge completed by three full rounds and one full fillet.
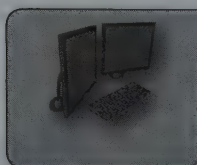
Exercise 8-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 8-4.

Figure 8-13.
A hinge created by adding three full rounds, one full fillet, and two holes to an extrusion. Add the holes using concentric placement after placing the full radii.

Full round

Full round

Full round

Full fillet
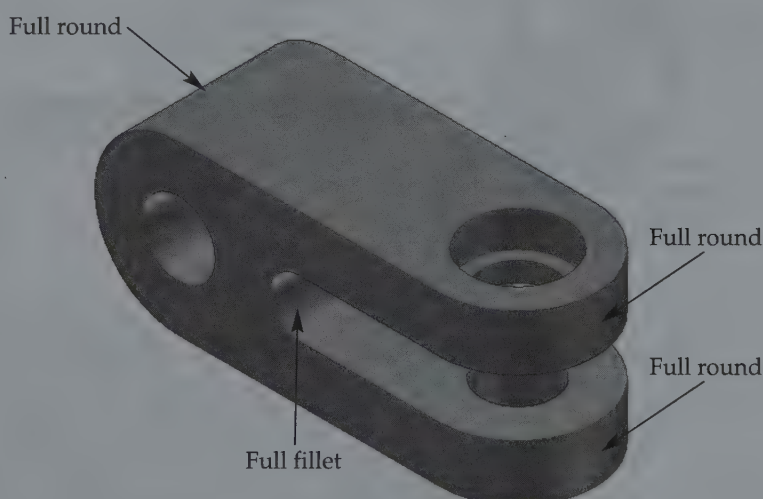
Figure 8-14.
The **Full Round Fillet** option allows you to create full radius fillets and rounds that can adapt to design changes.

Full round created using the **Full Round Fillet** option

Full round automatically adjusts to design changes

Original Extrusion

Revised Extrusion
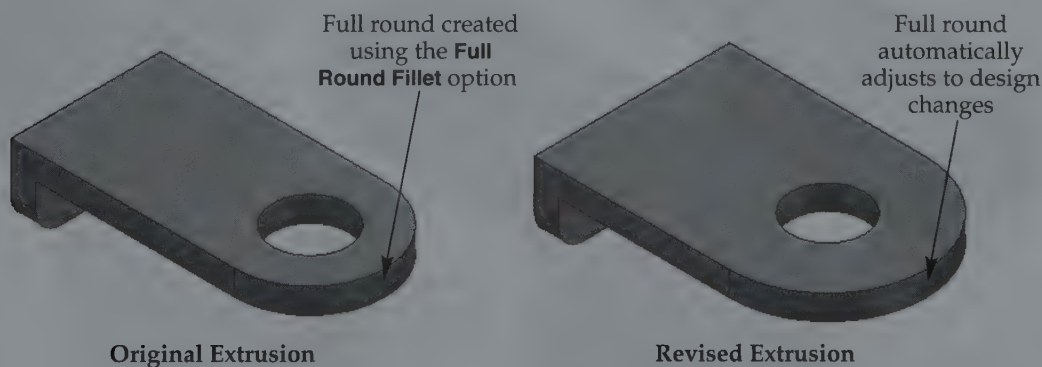
Chamfers

Chamfers relieve sharp edges, aid the entry of a pin or thread into a hole, ease inside corners, or replicate the use of an angle milling cutter. **Figure 8-15** shows an example of a part with multiple chamfers. You can place chamfers on selected edges or at the intersection of selected faces. You can apply chamfers at the corner of two intersecting, nonparallel planar faces. Access the **Chamfer** tool to create chamfers using the **Chamfer** dialog box. See **Figure 8-16**.

chamfer: Angled planar face placed on a feature edge

Ribbon
Model > Modify

Chamfer

NOTE

The **Chamfer** tool creates one or more chamfers of the same size. You must reuse the tool to create a chamfer of a different size. All chamfers placed during one operation make up a single feature in the browser.

Chamfer Options

Before you place chamfers, you should understand how the options in the **More** area of the **Chamfer** dialog box affect chamfer creation. Pick the **All tangentially connected edges** button to select a chain of tangent edges with a single pick. Pick the **Single Edge** button to pick individual segments. See **Figure 8-17**.

Pick the **Setback** button to apply setback when chamfering parts with three or more intersecting edges. Pick the **No Setback** button to provide no setback, which establishes a point at the intersection of the edges instead of a flat surface. See **Figure 8-18**. Pick the **Preserve All Features** check box when features intersect a chamfered corner. When this box is checked, existing feature geometry is not modified by a new chamfer that intersects the feature.

Figure 8-15.
Examples of several chamfers placed on a part.

Figure 8-16.
The **Chamfer** dialog box with the **Distance** option selected.

Pick to create a single-distance chamfer

Pick to create a distance-and-angle chamfer

Pick to create a chamfer using two distances

All tangentially connected edges button

Single edge button

Setback button
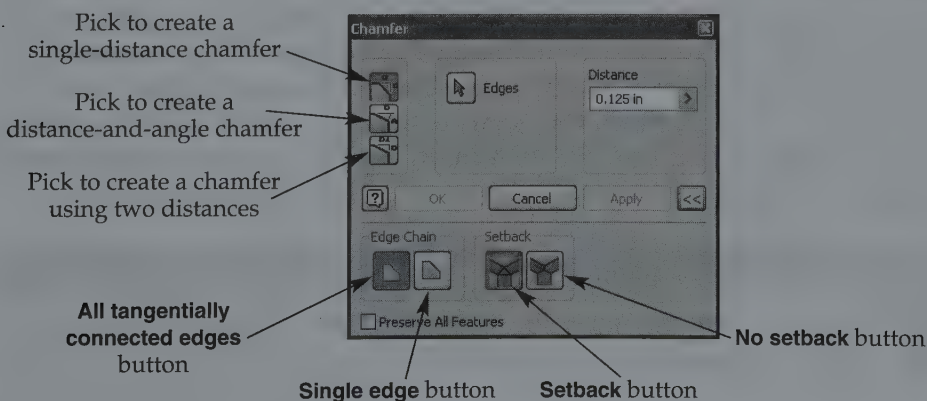
No setback button

Figure 8-17.
You can select edges for a chamfer by selecting an entire edge chain or an individual segment.

Individual edge piece

Entire edge chain

All Tangentially Connected Edges

Edge Chain

Single Edge

Edge Chain
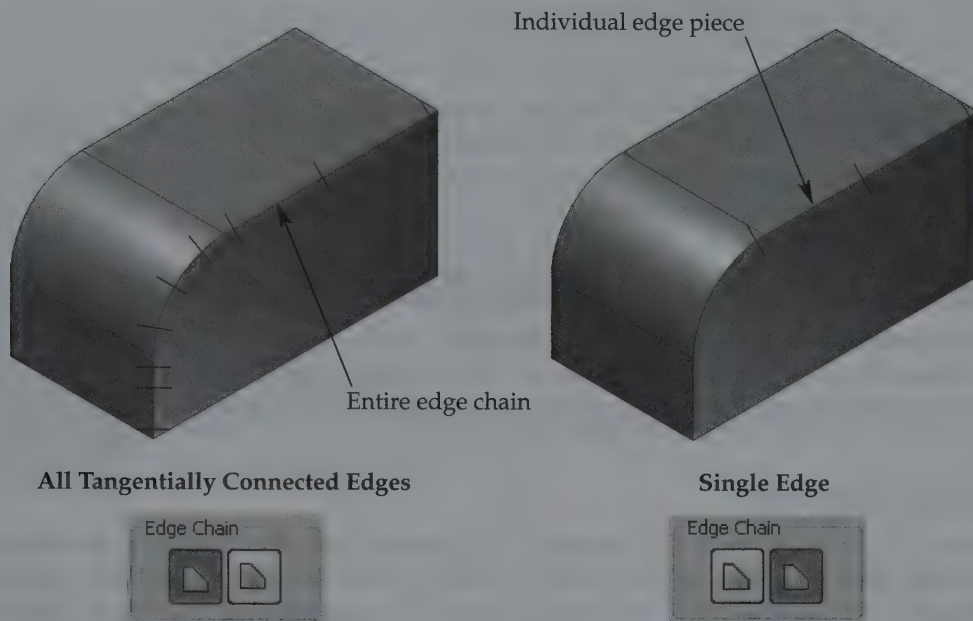
Figure 8-18.
An example of a chamfered corner with the **Setback** button selected and with the **No Setback** button selected.

Setback button selected

No Setback button selected

Placing Chamfers

To create a chamfer, select the appropriate chamfer method button. **Figure 8-19** shows how each method defines the size and shape of the chamfer. The **Distance** button is selected by default and allows you to create a chamfer with equal-distance sides from the selected edge, or a 45° angle. Pick the edges to chamfer and specify the chamfer distance using the **Distance** text box. Pick the **Apply** button to create the feature and remain in the **Chamfer** tool, or pick the **OK** button to create the feature and exit.

> **NOTE**
>
> Deselect selected edges by holding down [Ctrl] and picking the edges to remove from the operation.

Distance **Distance and Angle** **Two Distances**
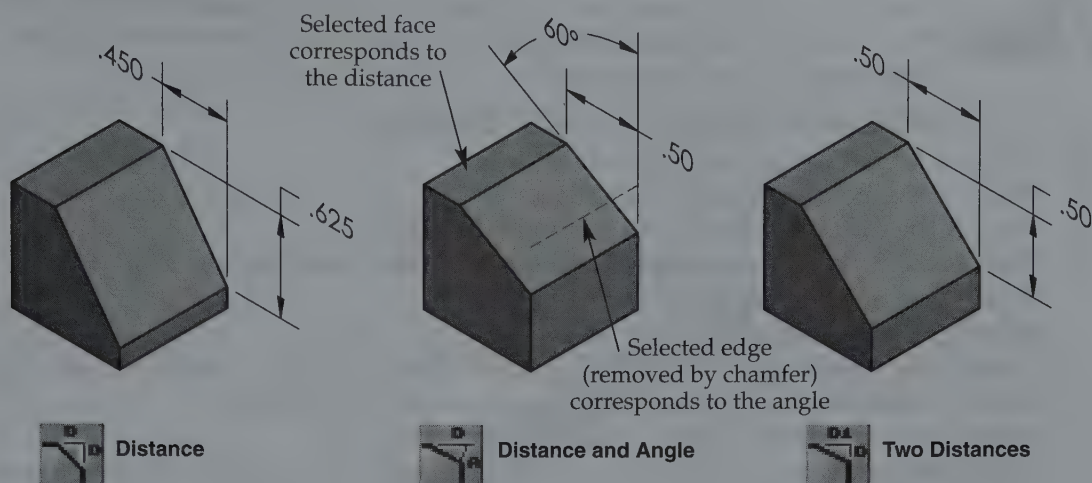
To specify one chamfer side as a distance from the edge and the other side as an angle from the edge, pick the **Distance and Angle** method button. The **Face** button is initially active, allowing you to select the face from which the chamfer distance is measured from the edge. The **Edge** button then activates, allowing you to select the edges from which the angle is measured. To change the face, pick the **Face** button again and reselect the edges. After you establish the face and edges to chamfer, specify a chamfer distance using the **Distance** text box and a chamfer angle using the **Angle** text box. Pick the **Apply** button to create the feature and remain in the **Chamfer** tool, or pick the **OK** button to create the feature and exit.

To specify one chamfer side with a distance from the edge, and the other side with a different distance from the edge, pick the **Two Distances** button. Select the edge to chamfer, and enter the chamfer distances in the **Distance1** and **Distance2** text boxes. If necessary, pick the **Flip** button to reverse the direction of the chamfer distances from the edge. Pick the **Apply** button to create the feature and remain in the **Chamfer** tool, or pick the **OK** button to create the feature and exit.
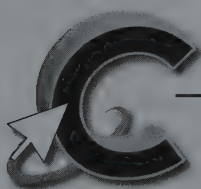
Exercise 8-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 8-5.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. What must be present before you can add a placed feature?
2. Name at least three types of placed features.
3. Briefly describe common uses for fillets.
4. What is the difference between a round and a fillet?
5. What is the function of the **Smooth radius transition** check box?
6. Briefly describe how to create a variable fillet.
7. What is a setback on a fillet or round?
8. Explain the effect when a setback is greater than the fillet or round radius, and when a setback is less than or equal to the fillet or round radius.
9. Briefly describe how to create a face fillet or round.
10. Which fillet tool allows you to create a radius that adapts automatically if you edit the underlying geometry?
11. Name at least three uses for chamfers in a model.
12. What is the purpose of the **All tangentially connected edges** button in the **Chamfer** dialog box?
13. List three different ways to create a chamfer using the **Chamfer** tool.
14. When selecting edges to chamfer, how can you deselect a single edge without leaving the **Chamfer** tool?
15. If you enter two distances for a chamfer, but the preview shows the opposite of the result you want, what can you do to correct it?

Problems

- *Open the specified part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **File:** P6-2.ipt
   Save as: P8-1.ipt
   Title: PIN
   Specific instructions: Place 2.375 mm rounds on both ends of the pin as shown.

2. **File:** P7-1.ipt
   Save as: P8-2.ipt
   Title: SWIVEL
   Specific instructions: Place a chamfer on the part as shown.

4.75

6.3
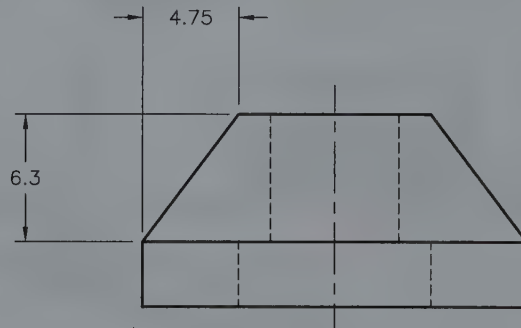
3. **File:** P7-2.ipt
   Save as: P8-3.ipt
   Title: BODY
   Specific instructions: Place 1.6 mm chamfers on the edges shown, 26 total with the **Single edge** button selected.

Chamfers

▼ Basic

▼ Basic

▼ Basic

4. **File:** P6-8.ipt
   Save as: P8-4.ipt
   Title: HANDLE
   Specific instructions: Use the **Fillet** tool and the **All Fillets** and **All Rounds** options to place .125″ fillets and rounds on all part edges.
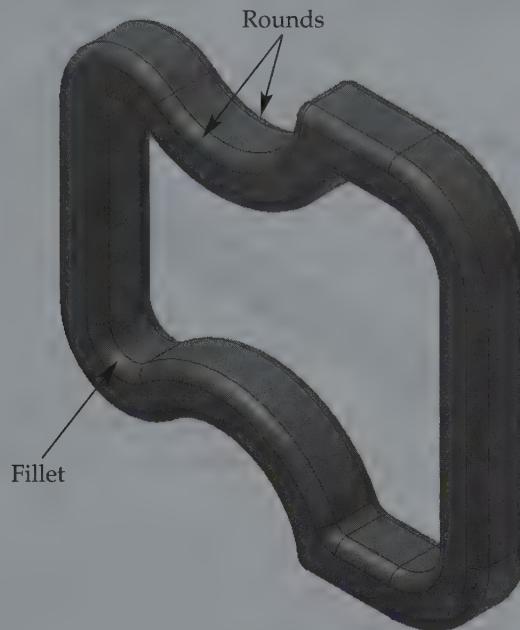
Rounds

Fillet

5. **File:** P7-4.ipt
   Save as: P8-5.ipt
   Title: SUPPORT BRACKET
   Specific instructions: Apply the fillets, rounds, and chamfers shown.

Fillet 1 edge
R .125″

Fillet 24 total selections
R .03125″

Fillet 1 edge
R .0625″

Chamfer 4 edges
.03125″
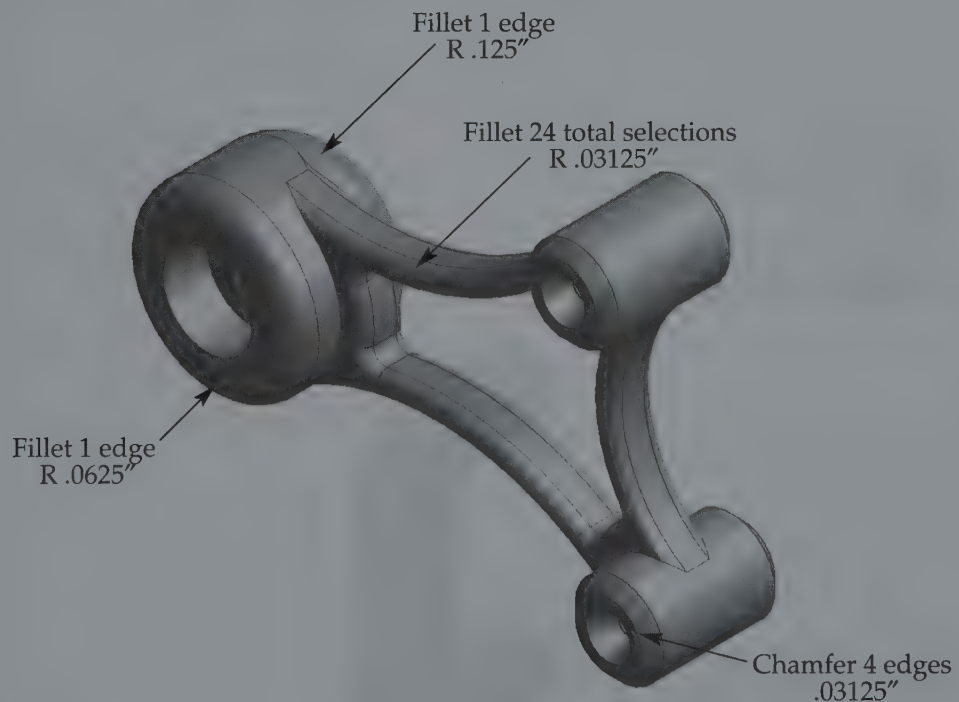
6. **File:** EX7-2.ipt
   Save as: P8-6.ipt
   Title: HITCH PIN BENT
   Specific instructions: Open a sketch on the XY plane. If you constructed the base feature sketch correctly, the sketch should be tangent to the bent cylinder. Sketch the **Point, Center Point** shown. To acquire the **Point, Center Point** location dimensional constraint value, right-click on the **Bend Part** sketch (possibly Sketch2) and select **Visibility**. Then, when placing the dimensional constraint, pick the existing .375 value used to locate the circle. Use the sketch to form a hole through the entire part. Reference the diameter of the .125 circle on the visible sketch to define the diameter of the hole. Turn off the visibility of the sketch. Add the .0625 chamfer and .125 round as shown. Add as much information as possible to the **iProperties** dialog box. Assign a Steel material and As Material color to the part.

Point, Center Point coincident to projected axis

fx:.375

Projected cylinder edge as a construction line style

Projected X axis is a construction line style

fx:.375

.0625 chamfer

.125 round

Problems – Chapter 8
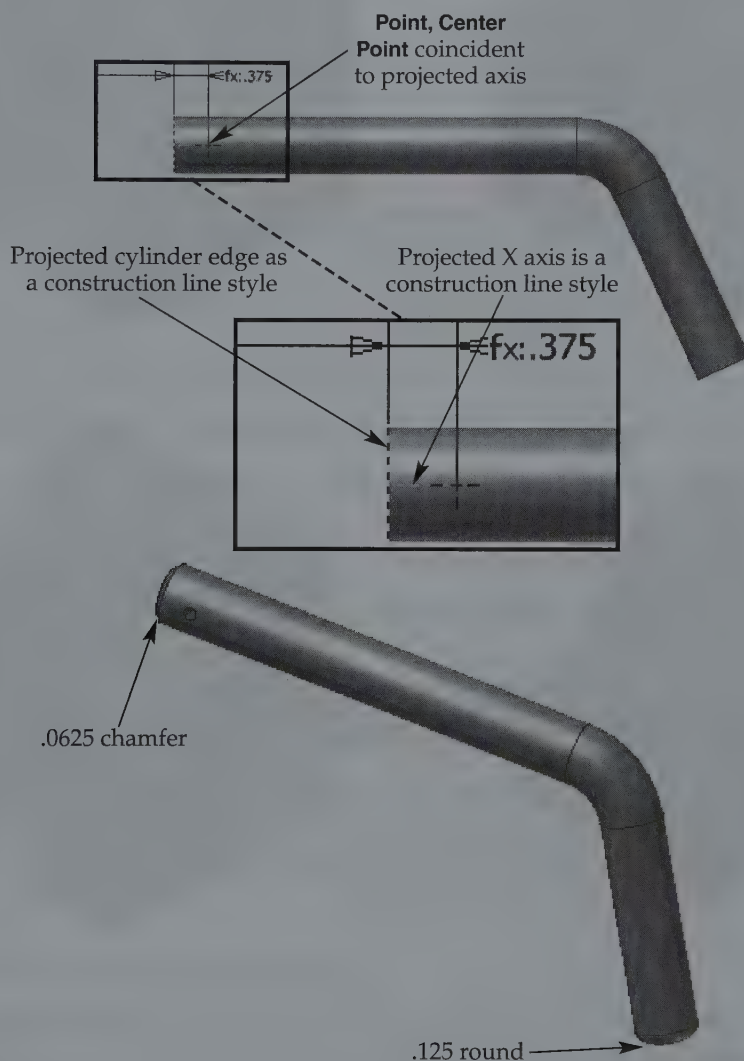
7. **File:** P6-1.ipt
   Save as: P8-7.ipt
   Title: SUPPORT BLOCK
   Specific instructions: Extrude the sketch 3″ in the positive direction as shown in A. Place a 2″ chamfer on the edges shown in B. Open a sketch on Face 1, and sketch the geometry shown in C. Add a .75″ hole through the entire part. The final part should look like the part shown in D.

Edges

A

Face 1

B

.500

Point, Center Point

.750

C

Hole through all

D
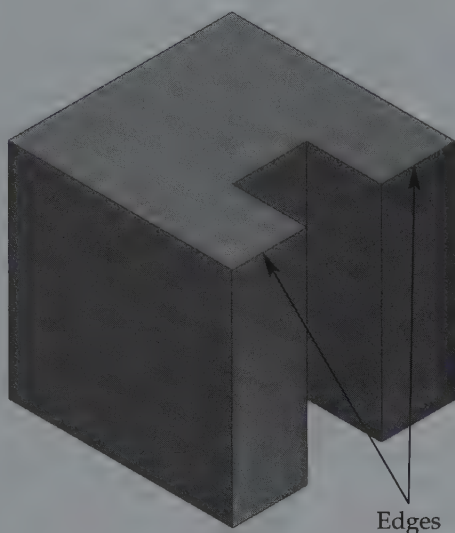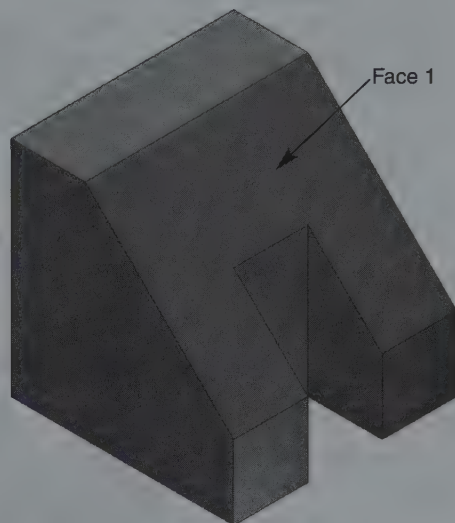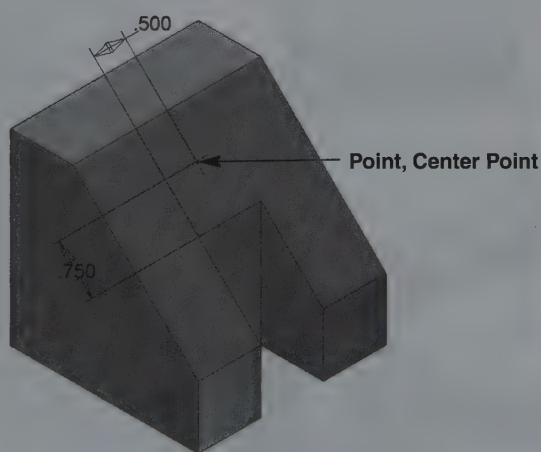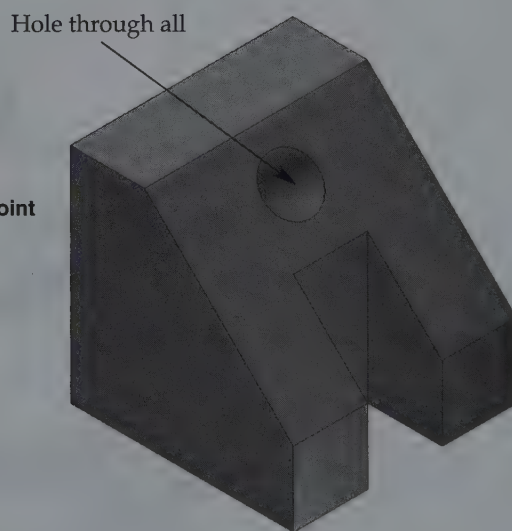
Additional Placed Features

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Apply face draft.
✓ Place threads on cylindrical and conical features.
✓ Shell features.
✓ Thicken faces and offset surfaces.
✓ Move faces.

This chapter describes common placed features that allow you to apply face draft, add threads, and shell features. You will also learn about placed features used to thicken faces, offset surfaces, and move faces. You can apply these features to parts for a variety of applications.

Face Draft

Face draft is often required to aid in removing a part or pattern from a mold, as shown in **Figure 9-1**. However, you can draft a face for any other application that requires a tapered face. Access the **Face Draft** tool to draft faces using the **Face Draft** dialog box. See **Figure 9-2**.

Using the Fixed Edge Option

The **Fixed Edge** button is active by default and allows you to identify the pull direction and the faces to draft using fixed edges. Use the active **Pull Direction** select button to select a face or plane to act as the *pull direction*. Refer again to **Figure 9-1**. An arrow identifies the direction from which the mold pulls. See **Figure 9-3**. If necessary, pick the **Flip** button to reverse the pull direction.

The **Faces** button activates, allowing you to select faces to draft. When you move the cursor over feature faces, a vector preview of the face draft operation appears, as shown in **Figure 9-3**. The vector indicates the fixed edge and face draft angle. When the vector displays the correct orientation, pick the face. Then set the draft angle using the **Draft Angle** text box. Pick the **Apply** button to create the feature and remain in the **Face Draft** tool, or pick the **OK** button to create the feature and exit.

face draft: A taper placed on a part surface.
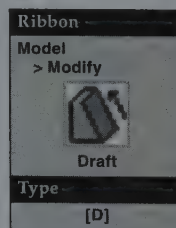
Ribbon

Model
> Modify

Draft

Type

[D]

pull direction: The direction in which a casting mold is pulled or removed from the part.

Figure 9-1.
An example of
applying face draft
to a mold and
molded part to aid
removing the part
from the mold.

Parting line

Molded part

Draft

Pull direction

Mold
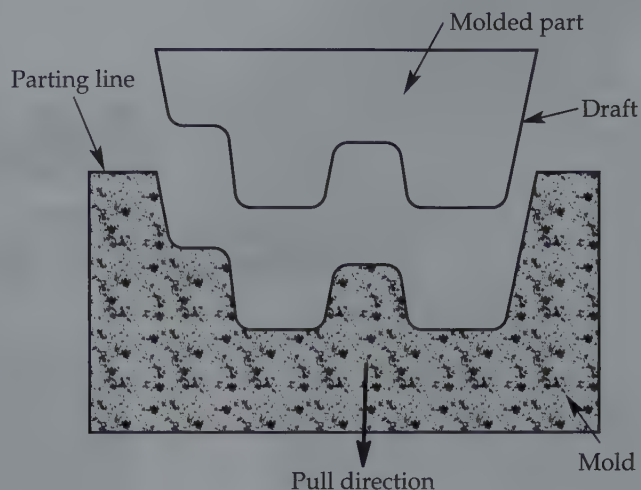
Figure 9-2.
A—The **Face Draft** dialog box with the **Fixed Edge** button selected. B—The **Face Draft** dialog box
with the **Fixed Plane** button selected.

Fixed
Edge
button

Fixed
Plane
button

Face Draft

Pull Direction

Draft Angle
2.5 deg

Faces

Ok   Cancel   Apply

A

Face Draft

Fixed Plane

Draft Angle
2.5 deg

Faces

Ok   Cancel   Apply

B
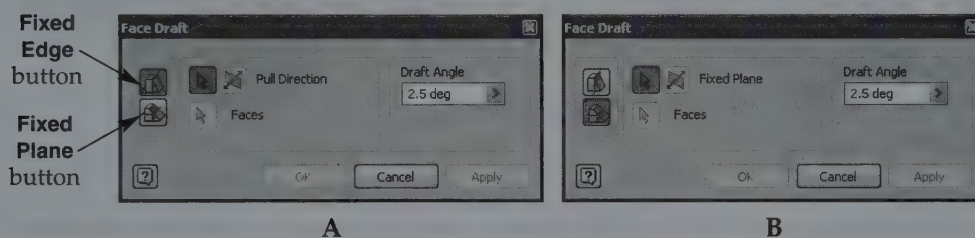
Figure 9-3.
Notice the position of the indicators during a fixed-edge face draft selection and the resulting part.

Pull direction
identification arrow

Selected pull
direction face

Face draft vector

Face draft vector

Face draft angle

Face draft angle

Selected face
fixed edge

Selected face
fixed edge

Before Drafting

After Drafting

Pull direction
identification arrow

Selected pull
direction face

Plane

Face draft
vector

Face draft angle

Before Drafting

After Drafting
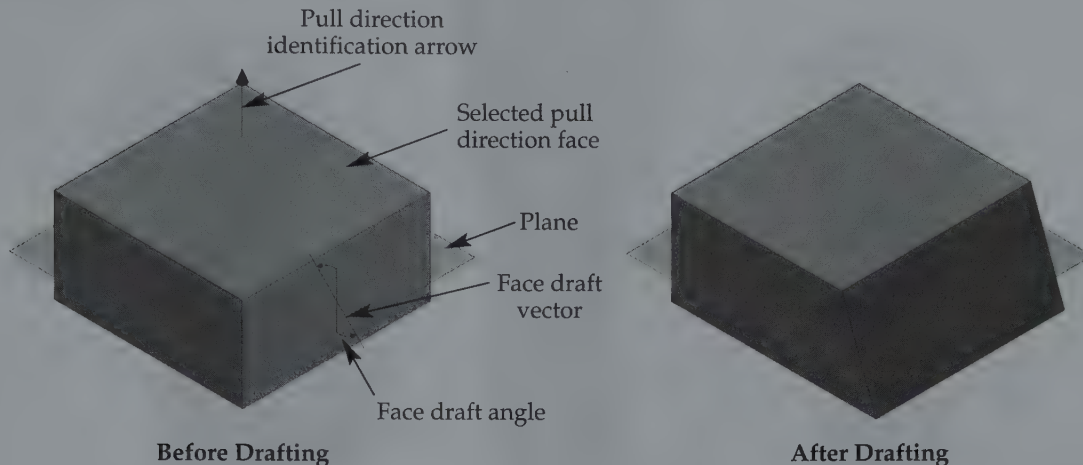
Using the Fixed Plane Option

Select the **Fixed Plane** button to define a fixed plane and the faces to draft in reference to the fixed plane. The main difference between the **Fixed Edge** and **Fixed Plane** options is that Inventor adds *or* removes material at the selected edge using the **Fixed Edge** function, but adds *and* removes material at the selected plane using the **Fixed Plane** function. The **Draft Plane** select button becomes active, allowing you to pick the pull direction by selecting a face or plane. For most applications, the plane you select should be a work plane or an existing feature face that will add and remove material during the face draft operation. See **Figure 9-4**. If necessary, pick the **Flip** button to reverse the pull direction.

The **Faces** button becomes active, allowing you to select faces to draft. When you move the cursor over feature faces, a vector preview of the face draft operation appears, as shown in **Figure 9-4**. The vector indicates the fixed edges and face draft angle. Notice that you can only define the face draft angle and fixed edges in relation to the selected fixed plane. Now set the draft angle using the **Draft Angle** text box. Pick the **Apply** button to create the feature and remain in the **Face Draft** tool, or pick the **OK** button to create the feature and exit.

Exercise 9-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 9-1.

Threads

For most applications, add *internal threads* to a hole using the **Hole** tool to increase productivity and reduce the number of model features. The **Thread** tool allows you to add *external threads,* or internal threads when the **Hole** tool is not appropriate or if you plan to suppress threads, but not the hole. **Figure 9-5** shows an example of parts with corresponding internal and external threads.

internal threads:
Thread forms on an internal hole feature.

external threads:
Thread forms on an external feature such as a pin, shaft, bolt, or screw.

External threads

Internal threads
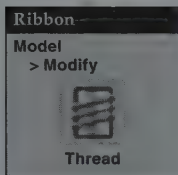
offset: When
referring to threads,
the distance from
the edge of the face
to the beginning of
the threads.

Access the **Thread** tool to display the **Thread** dialog box. See **Figure 9-6**. The **Face** button is active, allowing you to pick a cylindrical or conical face to apply threads. By default, a bitmap thread representation appears in the model. Deselect the **Display in Model** check box to hide the image. To thread the entire length of the selected face, pick the **Full Length** check box. To specify a certain length of threads, deselect the **Full Length** check box. Then specify the thread length using the **Length** text box and an *offset* distance using the **Offset** text box. See **Figure 9-7**. If necessary, pick the **Flip** button to reverse the offset.

Pick the **Specification** tab to specify the thread properties. Refer again to **Figure 9-6**. The thread properties available for placing threads are essentially the same as those offered for threading a hole using the **Hole** tool. Specific thread types, such as NPT, are available when you add threads to a conical feature to replicate tapered pipe threads. Only those designation options that correspond to the selected size are available. Pick the **Apply** button to create the feature and remain in the **Thread** tool, or pick the **OK** button to create the feature and exit.

Figure 9-6.
A—The **Location** tab of the **Thread** dialog box. B—The **Specification** tab of the **Thread** dialog box.

Deselect to specify thread length

Pick to flip the offset origin

Pick to specify thread properties

A

B
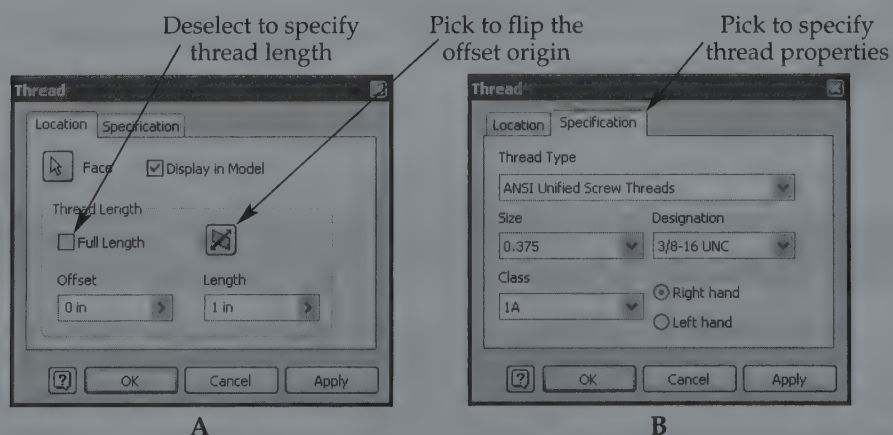
Figure 9-7.
Threads may cover the full length of a face, partial length without an offset, or partial length with an offset.

Full Length   **No Offset**   **With Offset**
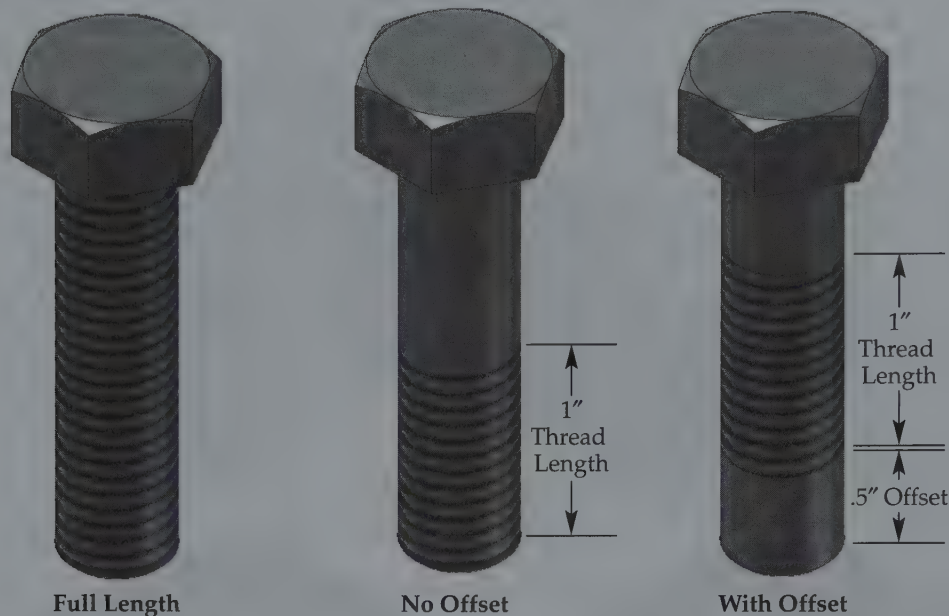
NOTE

Thread representations created using the **Thread** tool are fully parametric, and as a result, you can use model thread information to annotate drawings or for other applications. Thread representations are adequate for most models. Use the **Coil** tool to create actual detailed threads.

PROFESSIONAL TIP

A spreadsheet outlines the thread information available in the **Specification** tab. The location of the spreadsheet is Program Files\ Autodesk\Inventor 2010\Design Data\Thread.xls. Use this spreadsheet to modify and add thread types and thread information for specific applications. Changes made to the Thread spreadsheet do not alter existing thread specifications.
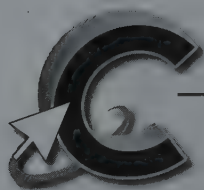
Exercise 9-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 9-2.

---

shell: An operation that removes material from a feature and creates a hollow space or opening.

A *shell* is a feature that allows you to remove model material and faces, leaving behind a specific wall thickness. See **Figure 9-8**. Common shell applications include cast or forged part designs and thin or hollow parts. One or more selected faces determine the material removed by a shell. Shells modify features by removing material and creating new faces on the inside, outside, or inside and outside of existing feature faces.

Access the **Shell** tool to create shell features using the **Shell** dialog box. See **Figure 9-9**. The **Remove Faces** button is active by default, allowing you to specify the faces to remove during the shelling process. You can remove one or more faces, as shown in **Figure 9-10**. The **Automatic Face Chain** check box is also active by default, allowing you to select multiple tangent faces in a single pick. See **Figure 9-11**. Select the **Inside** button to create shell walls on the inside of the existing feature faces. Pick the **Outside** button to create shell walls on the outside of the existing feature faces. Select

Ribbon

Model
> Modify

Shell

Figure 9-8.
An example of using a shell feature to develop a part.

Initial Features

Part after Shell
Is Applied

Final Part after Adding Holes
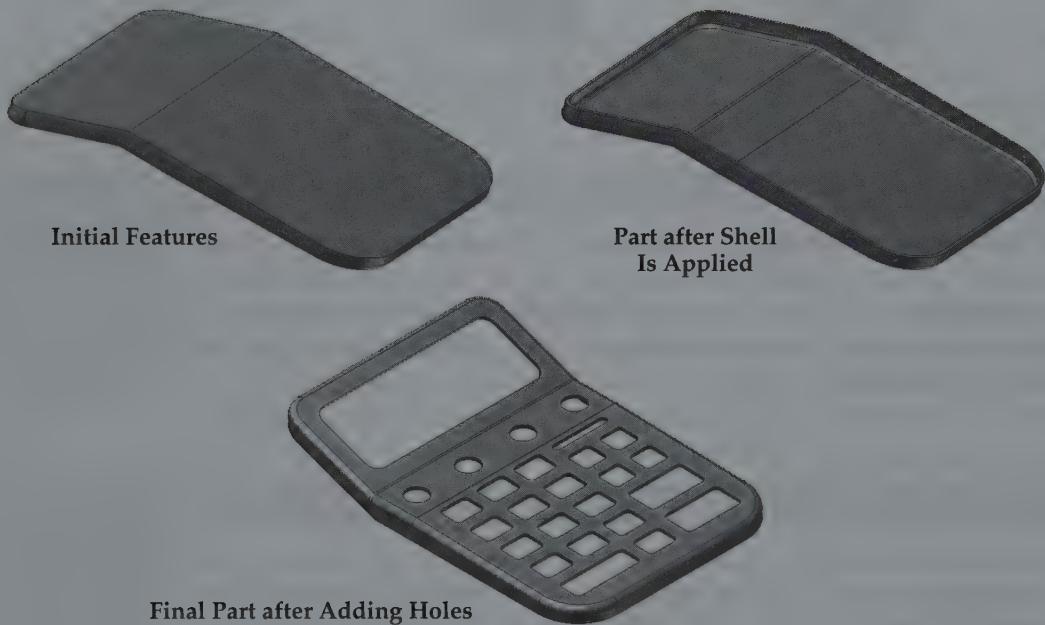
Figure 9-9.
The **Shell** dialog box.

Pick to shell inside of a selected face

Pick to shell outside of a selected face

Pick to shell both sides of a selected face
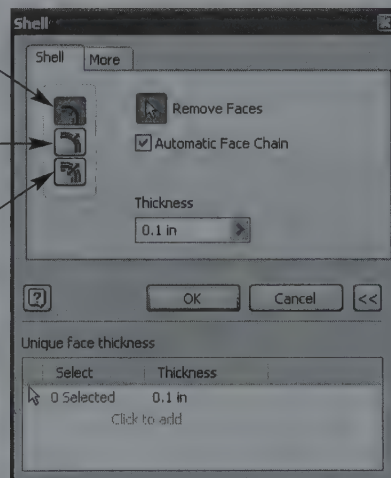
Figure 9-10.
Removing one or multiple feature faces during a shell operation.

Three feature faces
were removed

One feature face
was removed
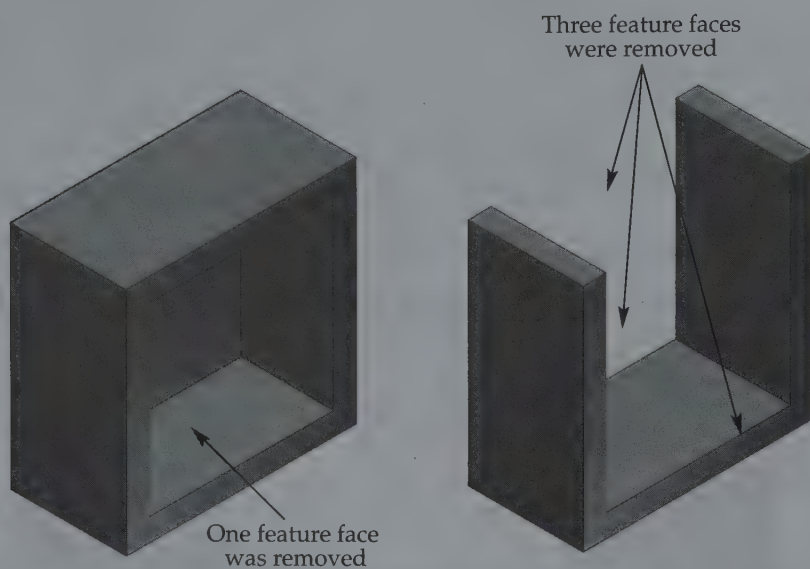
Figure 9-11.
Toggle the **Automatic Face Chain** check box on or off to select individual faces or a group of tangent faces. The **Split** tool, described in Chapter 13, creates the circular face shown.

Two faces
selected at once

Automatic Face Chain check box selected

Individual face
selected

Automatic Face Chain check box deselected
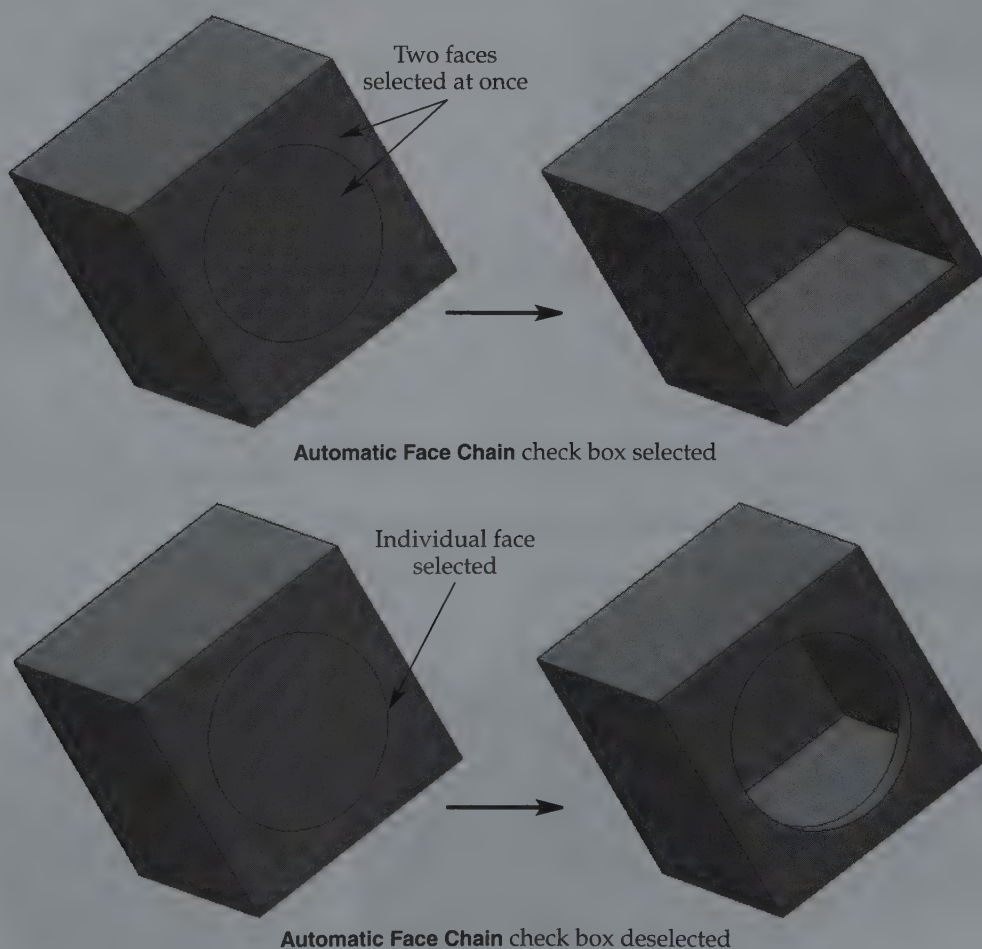
the **Both** button to create shell walls equally thick on the inside and outside of the existing feature faces. See **Figure 9-12**.

Next, specify the wall thickness, or the amount of material left after the shell operation, using the **Thickness** text box. The default shell applies a uniform thickness throughout the shell. To apply different thicknesses to certain walls, as shown in **Figure 9-13**, pick the **More** button to access the **Unique face thickness** area. Pick the **Click to add** button and select the faces to assign a unique wall thickness. Then specify a thickness using the **Thickness** text box. Add as many face thicknesses as required.

Figure 9-12.
The shell thickness can occur inside, outside, or inside and outside of the original part.

Original Part

Inside

Outside

Both
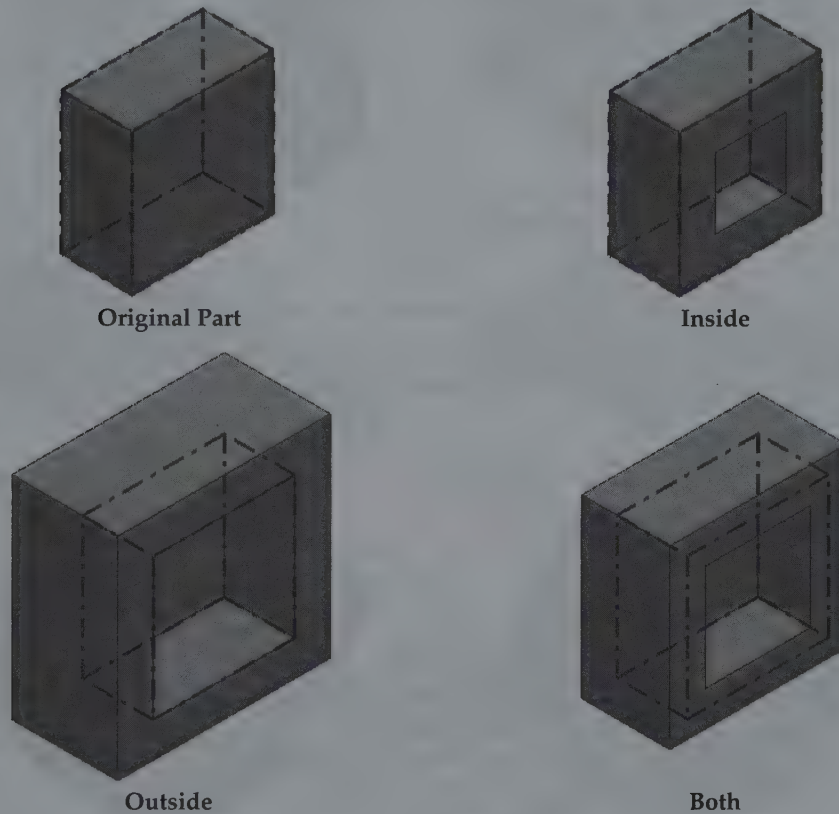
Figure 9-13.
Pick the **More** button to add unique face thicknesses to a shell feature.

More button

Existing Features
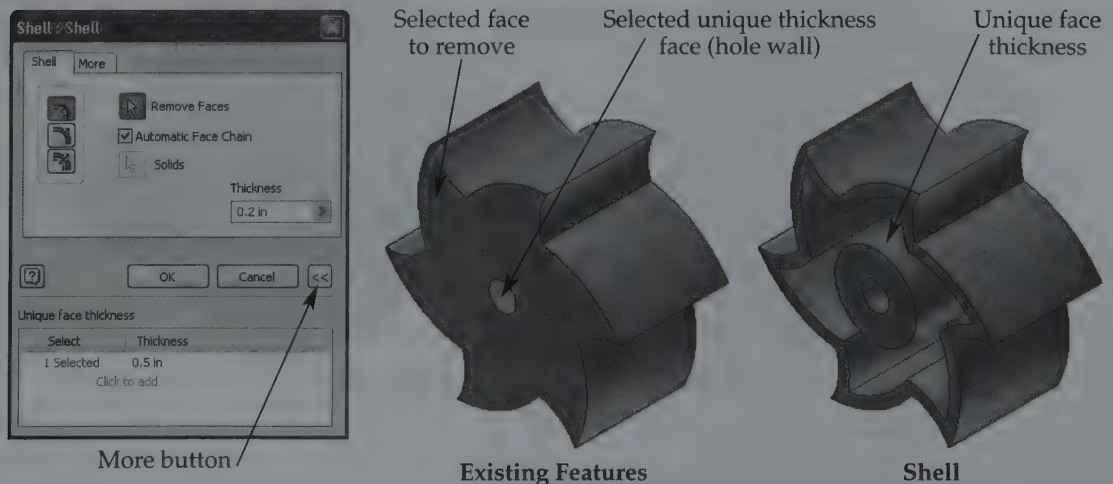
Shell

Controlling Approximation

A shell forms new faces parallel to existing faces. The new parallel faces, or offsets, are set according to the specified shell thickness. The offset is consistent throughout the shell except when the thickness is too large. **Figure 9-14** shows a shelled, sharp-pointed loft. Notice that the shell thickness is not consistent when it becomes too large to form a parallel surface. The **Allow Approximation** check box is selected by default and attempts to produce the shell by adjusting the thickness. To adjust for this issue, pick the **More** tab in the **Shell** dialog box. See **Figure 9-15**.

The drop-down list in the **Allow Approximation** area controls the approximate solution. Select the **Never too thin** option to increase the shell thickness as necessary, but never less than the specified thickness. Pick the **Never too thick** option to decrease the shell thickness as necessary, but never more than the specified thickness. Choose the **Mean** option to increase or decrease the thickness as needed. Pick the **Optimized** radio button to use an approximation tolerance that will result in the quickest shell, or choose the **Specify Tolerance** radio button to enter a tolerance. The approximation tolerance you enter in the text box is a percentage. A lower percentage results in higher approximation accuracy. Pick the **OK** button to create the shell.

> **NOTE**
>
> If you choose to approximate a shell solution, an alert box indicates that the offset is approximate. Pick the **Accept** button to produce the shell, or pick the **Edit** button to make modifications to the shell. Deselect the **Allow Approximation** check box to prevent approximation of shell thickness. This usually requires that you make significant changes to the model before creating the shell.

Exercise 9-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 9-3.

Figure 9-14.
An example of approximating shell offset when a consistent thickness is too large to form a parallel curved surface.
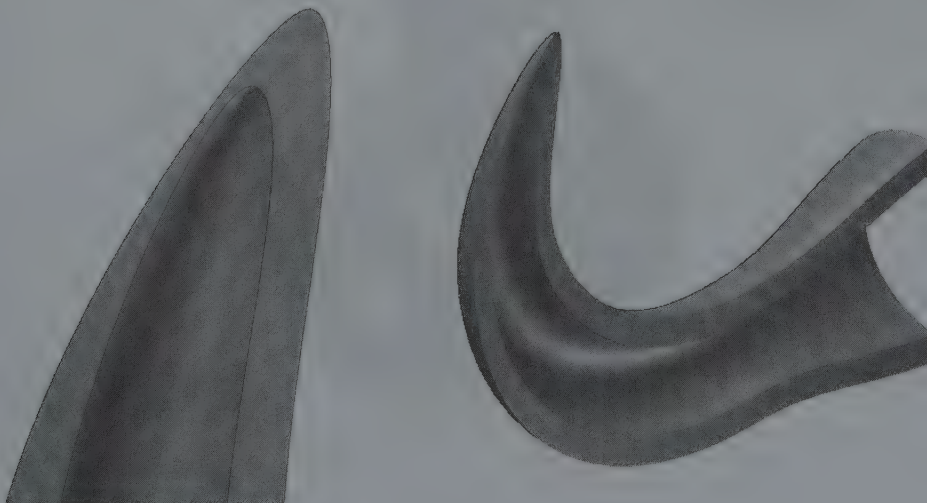
Figure 9-15.
The **More** tab of
the **Shell** dialog
box specifies
approximate shell
thickness.

Deselect
if thickness
approximation
is not a suitable
solution
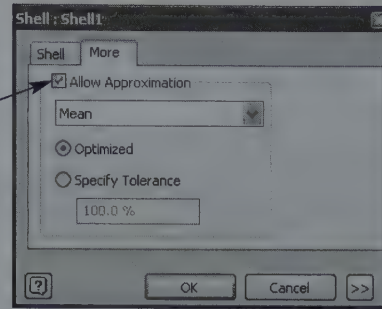
Thicken and Offset Features

A model face or surface may require additional material or an offset surface for construction purposes. Often the easiest, only, or most effective way to accomplish these tasks is to *thicken* or *offset* faces using the **Thicken/Offset** tool and corresponding dialog box. See **Figure 9-16**.

thicken: The process of adding a solid to a face or surface; similar to a solid extrusion.

offset: When referring to the **Thicken/Offset** tool, the process of offsetting a surface from a face or surface; similar to offsetting a work plane from a face.

Figure 9-16.
A—The **Thicken/Offset** dialog box. B—Examples of **Thicken/Offset** tool applications. You can easily create relatively complex geometry by thickening and offsetting existing faces.
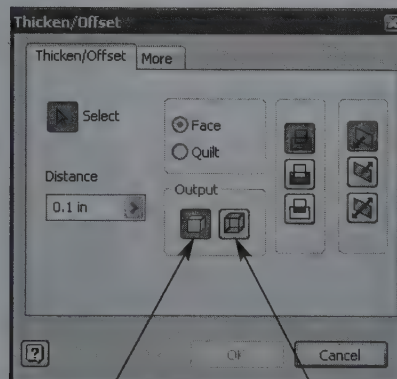
Pick to create a
solid thickening

Pick to create a
surface offset

A

Flange created by
thickening a fillet

Thickened
solid

Surface offset
from cylinder

Extrusions terminated
on offset surface

Fitting created by
thickening a chamfer face
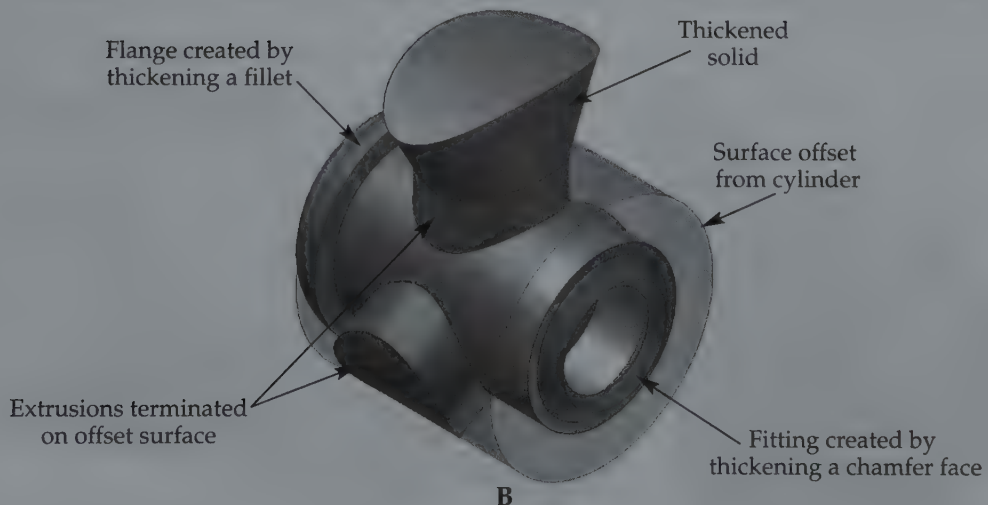
B

Pick the **Face** radio button to have selection access to faces or surfaces. Pick the **Quilt** radio button to have selection access to surfaces only. The **Select** button is active by default and allows you to pick feature faces or surfaces. Although you can choose multiple faces or surfaces, depending on the active radio button, you cannot pick faces and surfaces.

Pick the **Solid** button to generate a solid thickening, or pick the **Surface** button to create a volumeless surface offset. See **Figure 9-17**. Use the **Distance** text box to set the distance at which the thickened or offset feature occurs from the selected face or surface. You must specify a distance greater than 0 when thickening a feature, but you can enter any distance, including 0, when offsetting a surface. An offset distance of 0 is most common for creating a volumeless surface copy of existing solid faces for construction requirements of thin parts.

The **Operation** buttons are available only for thickening faces. These buttons allow you to choose whether the thickened face will join, cut, or intersect another feature. The **Direction** buttons control which side of the selected face or surface is thickened or offset.

The **More** tab, shown in **Figure 9-18**, controls additional thickening or offset characteristics. Pick the **Automatic Face Chain** check box to select multiple connected and tangent faces, instead of having to select each face individually. Pick the **Create Vertical Surfaces** check box to extend surfaces when offsetting external surfaces.

Ribbon
Model
> Surface
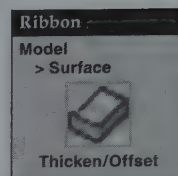
Thicken/Offset

Figure 9-17.
Notice the buttons from the **Output** area of the **Thicken/Offset** dialog box and the resulting geometry.

Thickened face

Offset surface

Existing Feature

Solid

Surface
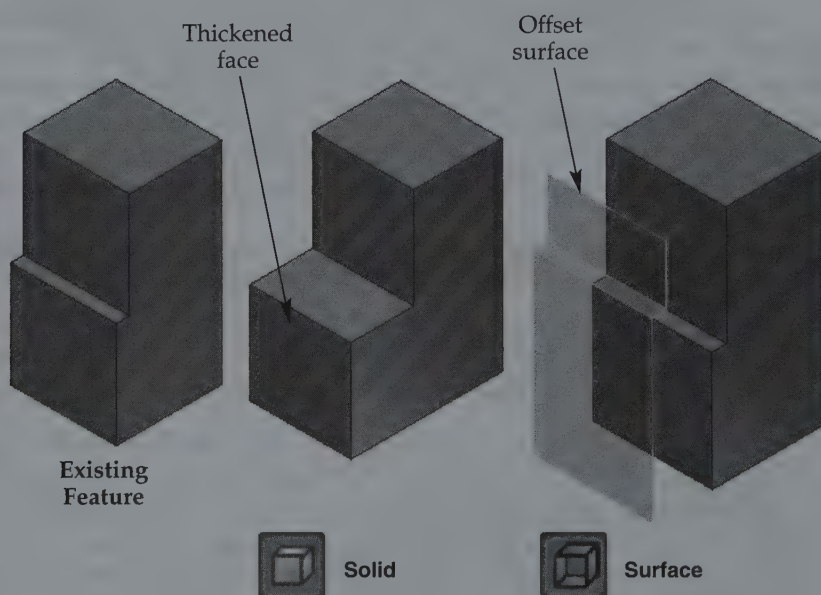
Figure 9-18.
The **More** tab of the **Thicken/Offset** dialog box contains an **Automatic Face Chain** check box that allows you to select multiple connected and tangent faces with one pick.
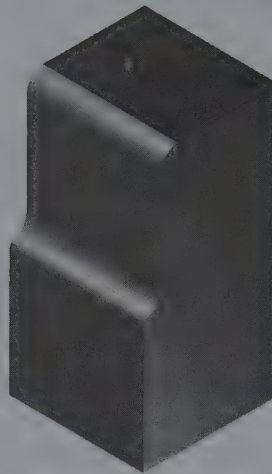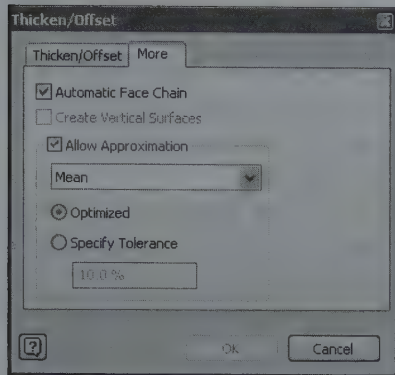
Automatic Face Chain
is unselected

Automatic Face Chain
is selected

Controlling Approximation

A thicken or offset forms new faces or surfaces parallel to the selected faces or surfaces. The new parallel faces or surfaces are set according to the specified thicken or offset distance. The distance is consistent throughout the operation except when the distance is too large. The **Allow Approximation** check box is selected by default and attempts to produce the thickening or offset by adjusting the thickness as needed. The approximation options are the same as those for the **Shell** tool. Pick the **OK** button to create the feature.

> **PROFESSIONAL TIP**
>
> In order to select an entire face chain, access the **More** tab of the **Thicken/Offset** dialog box first, pick the **Automatic Face Chain** check box, and then select the faces.

Exercise 9-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 9-4.

Moving Faces

If the size or shape of a feature requires modification, edit the feature parameters as needed. For example, to change an extrusion from 3″ to 2″, edit the extrusion distance using the **Extrude** dialog box. However, in some cases you may want to move a feature face without editing existing features. For example, you might want to explore a design option, provide needed clearance, or replicate a manufacturing process, such as creating a weld gap. Access the **Move Face** tool to remove material from or add to a selected face without eliminating existing feature parameters using the **Move Face** dialog box. See **Figure 9-19**.

Ribbon
Model
> Modify
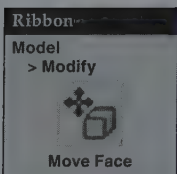
Move Face

Figure 9-19.
A—The **Move Face** dialog box. B—Moving a face using the **Move Face** dialog box and the **Direction Distance** option.

A

B
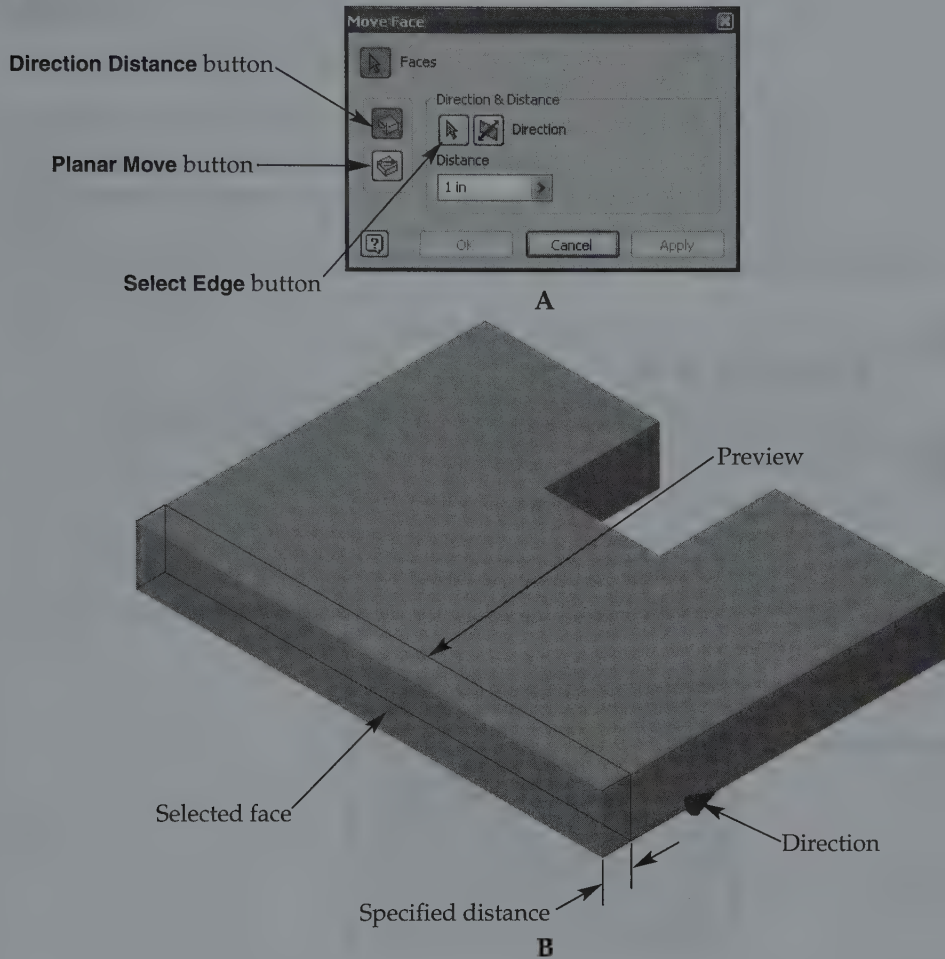
Using the Distance and Direction Option

The **Distance and Direction** button is active by default and allows you to move a face a specified distance along a linear edge. Use the **Select Faces** button to pick a face to move. Pick the **Select Edge** button and pick an edge to define where the face moves. For example, if you select a horizontal edge, the face moves along a horizontal plane. A preview arrow displays the direction and the intended effect of the move. If necessary, pick the **Flip Direction** button to move the face in the opposite direction. Specify how far to move the face using the **Distance** text box. Pick the **Apply** button to create the feature and remain in the **Move Face** tool, or pick the **OK** button to create the feature and exit.

Using the Planar Move Option

Pick the **Planar Move** button to move a face from one point to another on a specified plane. See **Figure 9-20**. The **Select Faces** button is active, allowing you to pick faces to move. After you select faces, pick the **Select Plane** button, located in the **Direction & Distance** area, and pick a plane to define where the faces move. Typically, you pick a plane that is not parallel to the selected faces.

Next, pick the **Select Points** button to define the move distance. The face moves from the first point to the second point you choose. You can select any available model points, such as endpoints or work points. You can also define the points using precise input techniques. Pick the **Apply** button to create the feature and remain in the **Move Face** tool, or pick the **OK** button to create the feature and exit.

Exercise 9-5

Figure 9-20.
A—The **Move Face** dialog box with the **Planar Move** option selected. B—Moving a face using the **Planar Move** option.
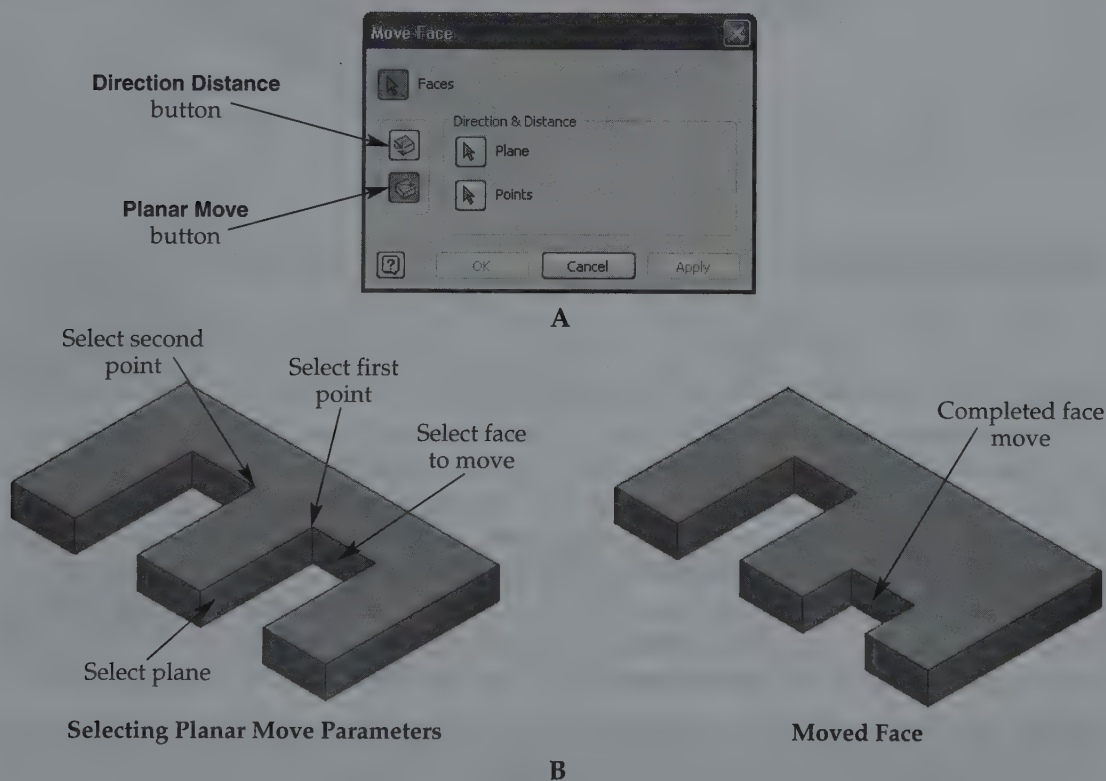
A

Selecting Planar Move Parameters

Moved Face

B

Chapter Test

1. What is a face draft?
2. Name two common uses for face drafts.
3. What is the pull direction in a face draft?
4. Explain the difference between internal and external threads.
5. Briefly explain when you should use the **Hole** tool to create threads and when you should use the **Thread** tool.
6. What does it mean to say that thread representations created using the **Thread** tool are fully parametric?
7. Briefly describe the function and uses of the **Shell** tool.
8. By default, what happens if the specified shell thickness is too large for the feature to which it is applied?
9. What is the difference between thickening and offsetting as they relate to the **Thicken/Offset** tool?
10. Name the tool that adds material to or removes material from a selected face without eliminating existing feature parameters.

Problems

1–3 Instructions:
- Open the specified part file, and save the file using the given name.
- Follow the specific instructions for each problem to create the features.

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **File:** P6-4.ipt
   Save as: P9-1.ipt
   Title: SCREW
   Specific instructions: Place a 2 mm fillet on the edge shown. Apply full-length, right-hand threads using the specifications shown.

2 mm fillet

M10X1.5 – 6G
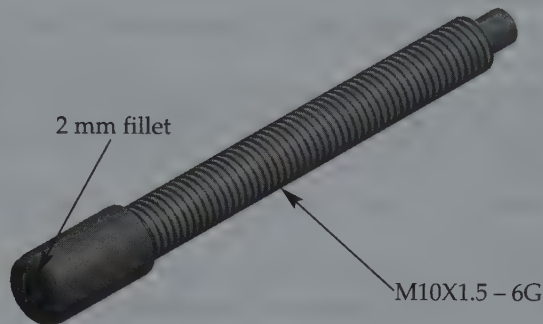
2. **File:** P5-10.ipt
   Save as: P9-2.ipt
   Title: SWIVEL BOLT
   Specific instructions: Thread the bolt stud using the specifications shown. The threads should be right-hand threads, .5" in length, and should have an offset of 0. Place a .025" chamfer on the edge shown.

.025" chamfer

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3. **File:** P6-10.ipt
   Save as: P9-3.ipt
   Title: FUNNEL
   Specific instructions: Place four .75" chamfers as shown. Shell the part using the following information:
   - Remove Face1 and Face2 using an **Inside** shell method and apply a thickness of .0625".
   - Apply a unique thickness set as the extrusion parameter, possibly d9 (the .125" extrusion value) to Face3 and Face4.
   - Place .03125" fillets and rounds using the **All Fillets** and **All Rounds** check boxes.

The final part should look like the part shown.

Chamfer
four
edges

Face1

Face2

Face3

Face4

4–5 Instructions:
- *Create sketches of the following objects*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

4. **Title:** HOUSING
   Units: Metric
   Template: Part-mm.ipt
   Part Number: IAA-013-01
   Project: REACTOR
   Material: Cast Steel
   Color: As Material
   Save as: P9-4.ipt
   Specific instructions: Open a sketch on the XZ plane. Extrude a 150 mm × 145 mm rectangle 50 mm in a positive direction. Sketch a ∅70 mm circle in the center of the top face of the extrusion. Extrude the circle 35 mm in the positive direction. Fillet the specified edges using the information provided. Shell the features using the following information:
   - Remove Face1 and Face2
   - Thickness: 10 mm
   - Direction: Inside

Sketch a 20 mm radius arc in the center of the face shown. Cut-extrude the arc using the **To Next** termination option. The final project should look like the part shown.

8 edges total
10 mm radius
Unselect **Rolling ball where possible** check box

Face2
(top cylinder face)

1 edge
5 mm radius

Face1
(bottom part face)
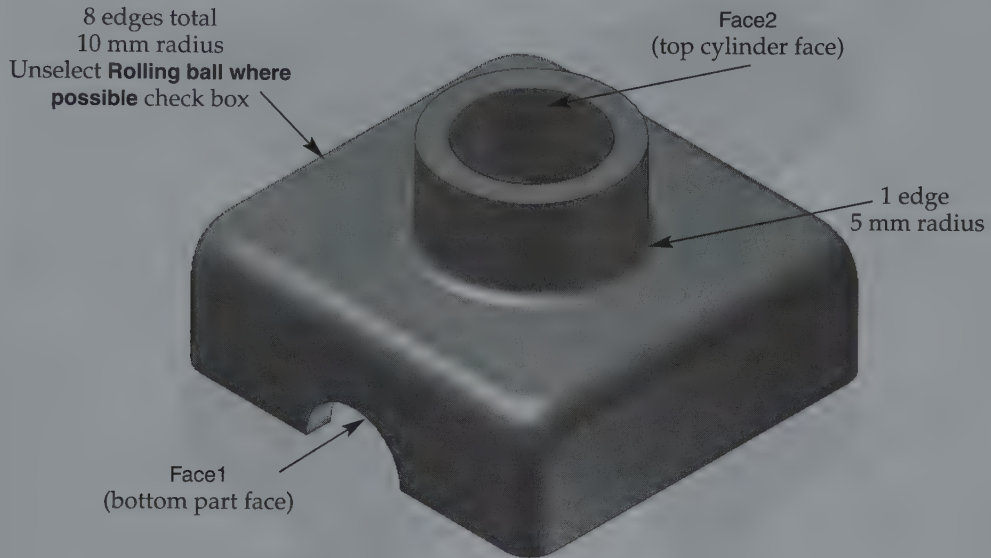
5. **Title:** BOTTLE
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-014-01
   Project: BOTTLE
   Material: Choose a material
   Color: Choose a color
   Save as: P9-5.ipt
   Specific instructions: Use the following tools to create a bottle similar to the bottle shown: 2 extrusions, constant fillets and rounds, variable rounds, 1 shell, and 1 threaded feature. Be sure to add the shell at the appropriate time, after the primary fillets and rounds, but before the threads and secondary fillets and rounds.

Feature Patterns

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Mirror features.
✓ Create rectangular feature patterns.
✓ Develop circular feature patterns.
✓ Work with feature patterns.

Use *feature pattern* tools to create mirrored features and circular or rectangular patterns without adding time-consuming sketch geometry or recreating features using conventional methods. Feature patterns consist of selected features and *pattern occurrences*. Pattern all features to be included in the pattern in a single operation. For example, pattern an extrusion and a chamfer and hole added to the extrusion at the same time. In order to pattern placed features, you must also select the features on which the placed features are located.

feature pattern:
An arrangement of features in a specific configuration created using feature pattern tools.

pattern occurrences:
Representations of patterned features that identify how many features are present because of the pattern operation.

Mirrored Features

Mirrored features are reversed, or mirrored, feature occurrences that are appropriate for symmetrical parts, or parts with symmetrical features. See **Figure 10-1**. Access the **Mirror** tool to mirror features using the **Mirror** dialog box. See **Figure 10-2**.

The **Mirror individual features** button is active by default for mirroring selected features. Use the **Features** button to select features to mirror. You may need to pick certain features from the browser if you cannot access them from the graphics window. Then pick the **Mirror Plane** button and select the *mirror plane*. Choose any available plane, including a planar feature face, work plane, or a plane in the **Origin** folder of the browser.

Pick the **Mirror a solid** button to mirror the complete part at the current design stage without selecting individual features. The **Mirror Plane** button is active, allowing you to select the mirror plane. The **Mirror a solid** option selects all solid features in the part. To include work features and surfaces in the mirror operation, pick the **Include Work/Surface Features** button and select work features and surfaces to mirror. If necessary, select the **Remove Original** check box to replace the parent features with the mirrored features, without actually deleting the parent features. Pick the **Join** button to add

mirrored features:
A mirror image of an existing feature created symmetrically over a specified plane.

mirror plane: A plane of symmetry about which features are mirrored.

Ribbon
Model
> Pattern

Mirror

Type
[Ctrl]+[Shift]+[M]
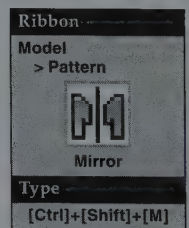
Figure 10-1.
An extrusion, hole, chamfers, fillets, and rounds mirrored twice to create a part.

Mirror plane (not shown)

Mirror plane

Initial Feature
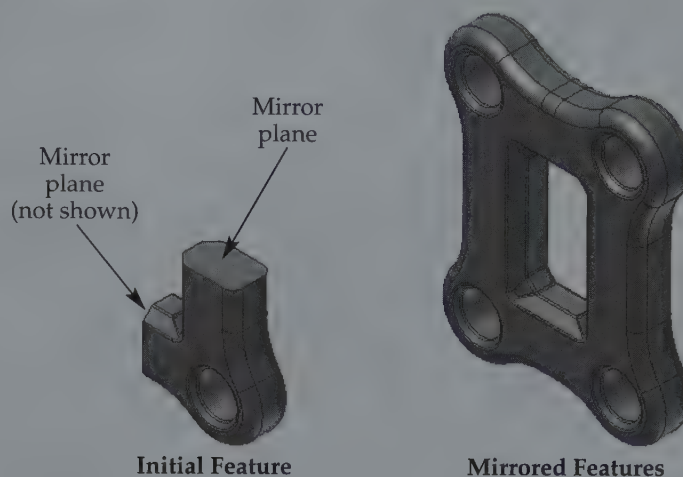
Mirrored Features

Figure 10-2.
The **Mirror** dialog box shown with the **Mirror individual features** or the **Mirror a solid** button selected.

Mirror individual features button

Mirror a solid button

Select for most applications when creating a single solid body part

Select only if you want to create a separate solid body

Mirror Individual Features

Mirror Entire Solid
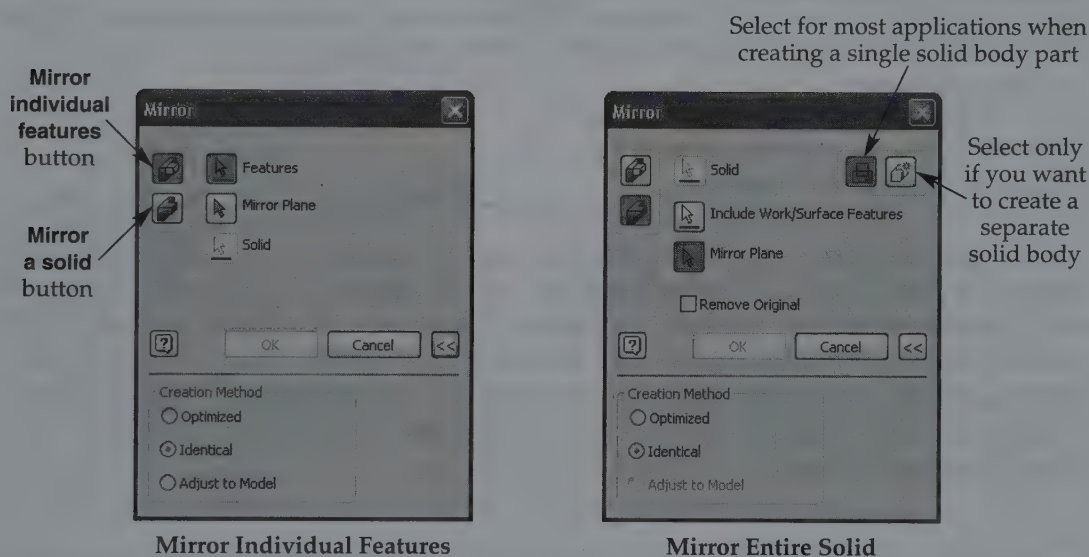
material to the selected solid, and disregard the **New solid** button unless you want the mirror to form a separate solid body.

A preview appears when you select the items to mirror and pick a mirror plane. To finalize the mirror, choose the **More** button to access the **Creation Method** area. The **Identical** radio button, selected by default, creates mirrored features that are the same as the parent features. All characteristics of the parent features, including the extents, are reproduced in the mirror. Use the **Identical** option when mirrored features terminate on a work plane or the same planar face. Select the **Optimized** radio button to create a mirrored copy of the parent feature's shape, but reproduce the features as faces, rather than as the features formed when you use the **Identical** method. All other characteristics of the parent features, including the extents, are reproduced in the mirrored faces.

Pick the **Adjust to Model** radio button to create mirrored features based on the parent features, but change the size to adjust for variations in the model, especially when the parent feature terminates on a feature face. **Figure 10-3** shows the difference between using an **Optimized** or **Identical** method and the **Adjust to Model** method. Pick the **OK** button to mirror the features.

Figure 10-3.
Mirroring a feature using the **Optimized** or **Identical** creation methods and the **Adjust to Model** creation method.

Parent feature

Mirrored feature is identical to the parent feature

Mirrored feature adjusts to changes in the model

Mirror plane

Optimized or **Identical** creation methods

Adjust to Model creation method
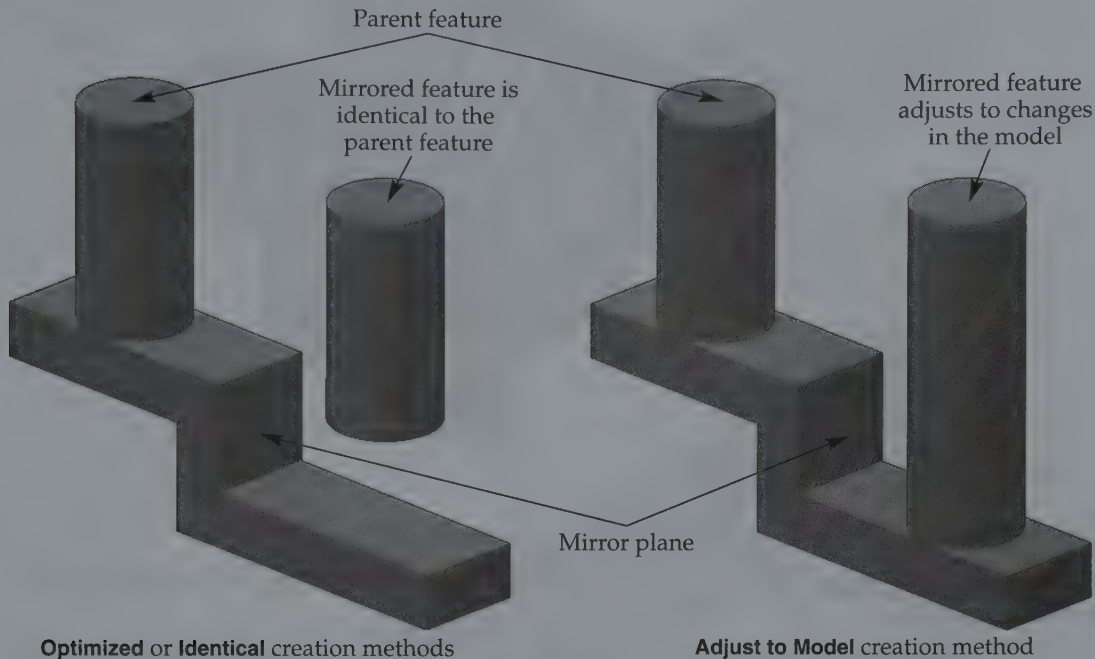
PROFESSIONAL TIP

The **Adjust to Model** creation method takes longer to generate mirrored features than the **Optimized** or **Identical** method. Select the **Adjust to Model** radio button only when mirrored features must be modified to fit changes in model geometry.
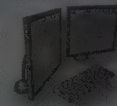
Exercise 10-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 10-1.

Rectangular Feature Patterns

A *rectangular feature pattern* allows you to produce a rectangular group of matching features efficiently. See **Figure 10-4**. Access the **Rectangular Pattern** tool to create a rectangular feature pattern using the **Rectangular Pattern** dialog box. See **Figure 10-5**.

The **Pattern individual features** button is active by default for patterning selected features. Use the **Features** button to select features to pattern. You may need to pick certain features from the browser if you cannot access them from the graphics window. An alternative is to pick the **Pattern a solid** button to select all solid features in the part. To include work features and surfaces in the pattern operation, pick the **Include Work/ Surface Features** button and select work features and surfaces to pattern. Pick the **Join** button to add material to the selected solid, and disregard the **New solid** button unless you want the pattern to form separate solid bodies.

> **rectangular feature pattern:** Occurrences of features copied and positioned a specified distance apart in rows and columns.

> **Ribbon**
> **Model**
> > Pattern
>
> **Rectangular Pattern**
> **Type**
> [Ctrl]+[Shift]+[R]
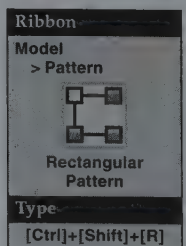
Figure 10-4.
This part includes
three rectangular
feature patterns:
a pattern of holes
and two separate
patterns of cut
extrusions.

Rectangular
patterns
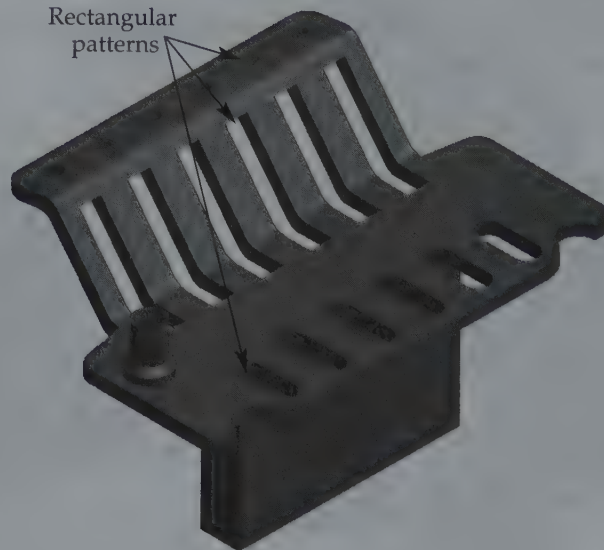
Figure 10-5.
The **Rectangular Pattern** dialog box with the **Pattern individual features** or the **Pattern a solid**
button selected.

Midplane check box

Pick before selecting the second direction

Pick to flip direction

Select for most applications when creating a single solid body part

Pattern individual features button

Pattern a solid button

Pick before selecting the first direction

Select only if you want to create a separate solid body

Pattern Individual Features

Pattern Entire Solid
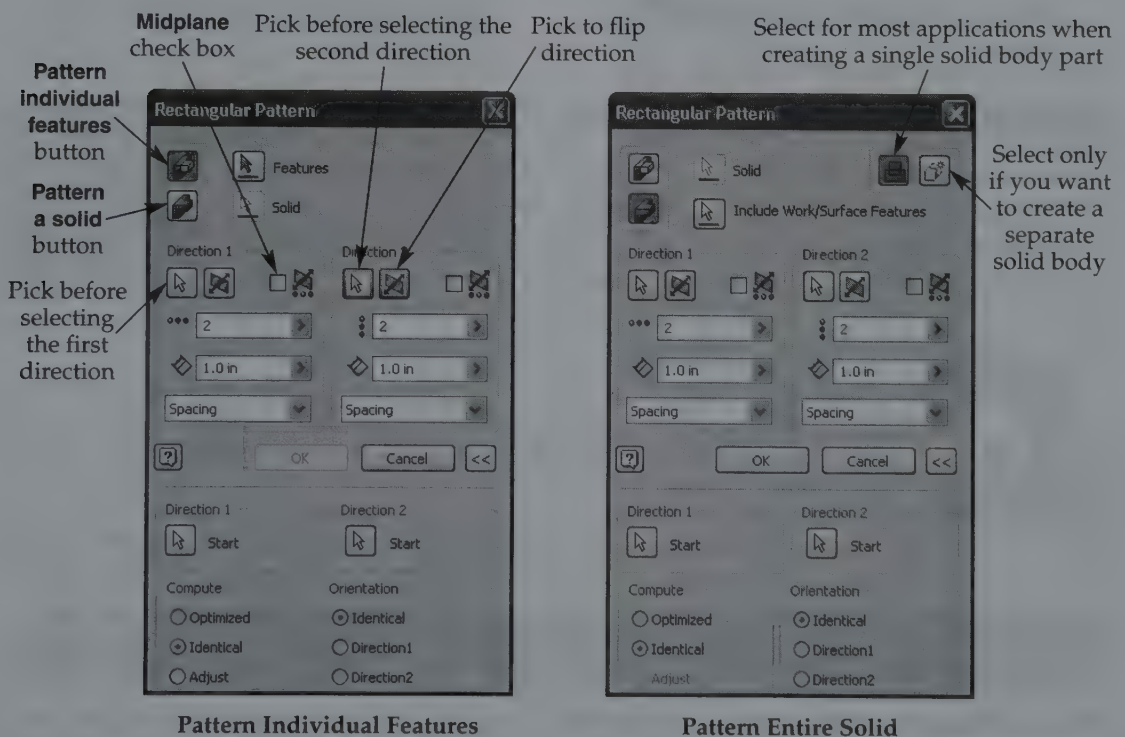
Use the **Direction 1** button to select the first pattern direction. You can pick a
feature edge, axis, or work axis, including the axes located in the **Origin** folder of the
browser. The path is parallel to a straight selection or tangent to a curved selection.
Another option is to pick a plane or face to specify the path perpendicular to the plane.
If the preview and direction arrow appear incorrect, pick the **Flip** button to reverse the
direction. Patterns form in a direction on one side or the other of the selected features,
depending on the flip direction. An alternative is to pick the **Midplane** check box to
spread the occurrences on each side of the selected features. You may still need to pick
the **Flip** button if you enter an even number of total features. See **Figure 10-6**.

Figure 10-6.
Using the **Midplane** direction option to create a rectangular pattern.

The total number of pattern features, 5 in this example, is divided on both sides of the selected item

This example shows how an even number of features may require flipping to create the correct pattern
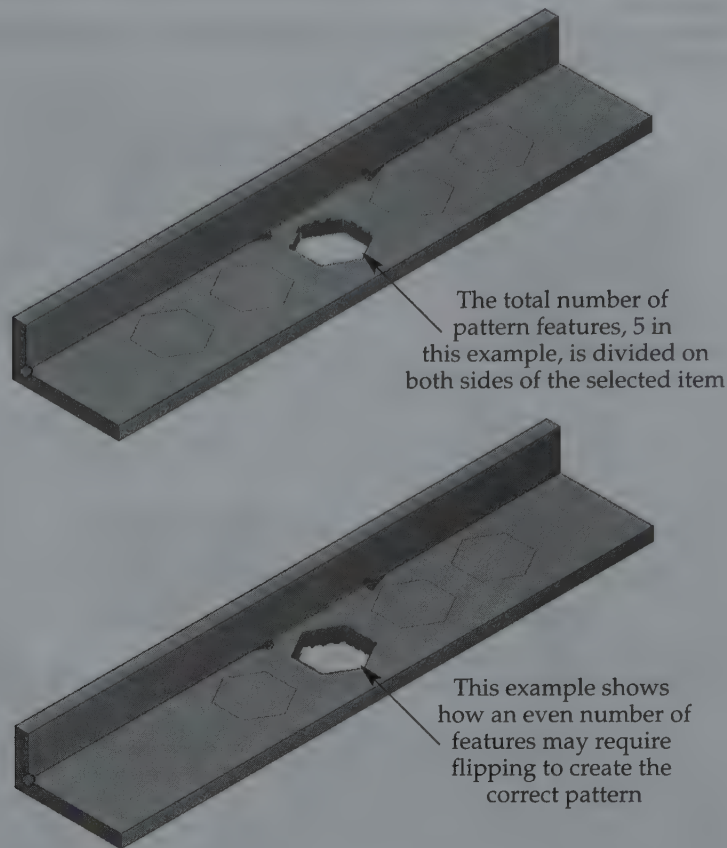
Specify the number of occurrences using the **Count** text box. Then define the distance between the copies using one of the options available from the drop-down list. The **Spacing** option is the default, allowing you to specify the *spacing* using the **Length** text box. For example, to pattern an extrusion that is 1″ (24.5 mm) wide and leave a 1/2″ (12.25 mm) space between copies, specify a 1.5″ (36.75 mm) spacing.

Select the **Distance** option to divide the count equally along the value you specify in the **Length** text box. Distance is measured from a point on the first pattern occurrence to the corresponding point on the farthest pattern occurrence. For example, the length between the center of the first cylinder in a pattern to the center of the last cylinder in a pattern defines the distance. The **Curve Length** option is available when you pick a feature curve, most often an edge, as the path. Use the **Curve Length** option to calculate and add the entire length of the curve to the **Length** text box.

After you define **Direction 1**, repeat the steps to define **Direction 2**. For most applications, the second direction is not parallel to the first direction. Direction arrows and a preview display the complete pattern operation. Additional pattern options are available in the lower portion of the dialog box. By default, the **Direction 1** and **Direction 2** patterns start at the beginning of the selected paths in reference to the specified direction. To change where the patterns begin, pick the **Direction 1** and **Direction 2** buttons and choose new points on the selected path curves.

The **Compute** area contains the same **Optimized, Identical,** and **Adjust** radio buttons available for the **Mirror** tool. The **Orientation** area contains options for defining rectangular feature pattern occurrences over complex curves. Pick the **Identical** radio button to create each pattern occurrence identical to the parent feature, without compensating for a complex curve. To adapt the pattern to a complex curve by rotating each pattern occurrence along the curve, pick the **Direction1** radio button for the **Direction 1** path, and the **Direction2** radio button for the **Direction 2** path. See **Figure 10-7**. Pick the **OK** button to pattern the features.

spacing: In patterning, the distance between occurrences based on the width of the selected features and the distance between the copies.

Figure 10-7.
A rectangular pattern of features with the **Identical** and **Direction1** or **Direction2** radio buttons selected (one direction shown).

Parent feature

Selected path curve

Parent feature

Selected path curve
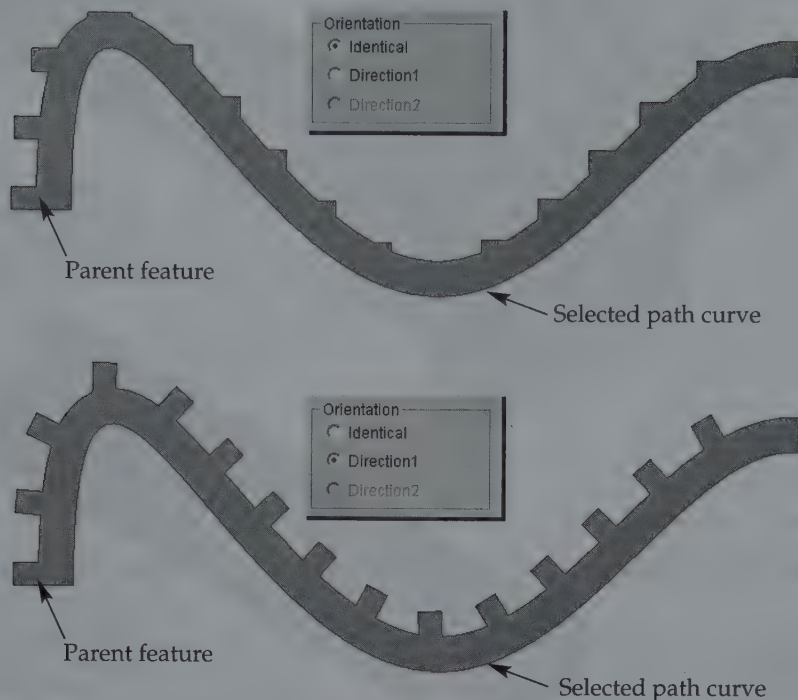
Exercise 10-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 10-2.

Circular Feature Patterns
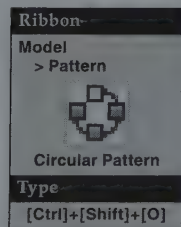
A *circular feature pattern* allows you to create a circular group of matching features efficiently. See **Figure 10-8**. Access the **Circular Pattern** tool to create a circular feature pattern using the **Circular Pattern** dialog box. See **Figure 10-9**.

The **Pattern individual features** button is active by default for patterning selected features. Use the **Features** button to select features to pattern. You may need to pick certain features from the browser if you cannot access them from the graphics window. An alternative is to pick the **Pattern a solid** button to select all solid features in the part. To include work features and surfaces in the pattern operation, pick the **Include Work/Surface Features** button and select work features and surfaces to pattern. Pick the **Join** button to add material to the selected solid, and disregard the **New solid** button unless you want the pattern to form separate solid bodies.

Use the **Rotation Axis** button to select the *axis of rotation*. You can select any feature edge, cylindrical surface, axis, or work axis available in the model, including an axis located in the **Origin** folder of the browser. If the preview and rotation arrow appear incorrect, pick the **Flip** button to reverse the rotation.

Next, specify the total number of occurrences using the **Count** text box. Before specifying the angle, pick the **More** button to access additional pattern options. The

Figure 10-8.
An example of a part built efficiently using a circular feature pattern.

Original Extrusion   Circular Pattern   Final Part with
                                                                          Additional Features

Figure 10-9.
The **Circular Pattern** dialog box with the **Pattern individual features** or the **Pattern a solid** button selected.

Pattern Individual Features   Pattern Entire Solid
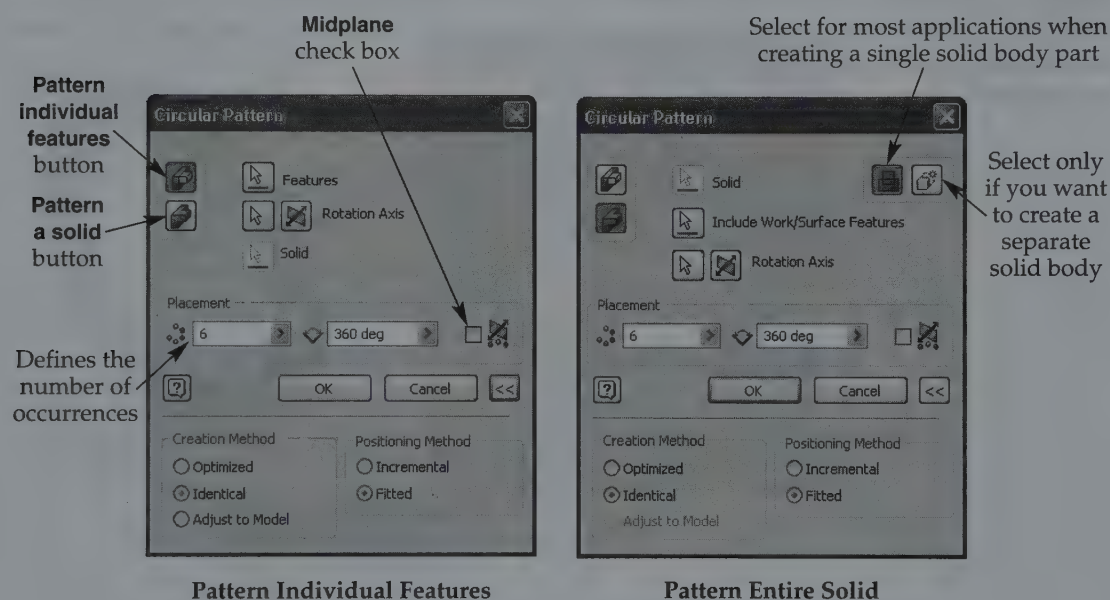
Compute area contains the same **Optimized, Identical,** and **Adjust** radio buttons available for the **Mirror** tool. The **Positioning Method** area includes options that control how the pattern forms. The **Fitted** radio button is selected by default, allowing you to divide the count equally inside the included angle specified in the **Angle** text box. A 360° angle is a full rotation. **Figure 10-10** shows an example of patterning a feature using a count of 6 and a 90° fitted angle. Pick the **Incremental** radio button to specify the included angle between each occurrence using the **Angle** text box. **Figure 10-11** shows an example of patterning a feature using a count of 6 and a 30° incremental angle.

If you are using either the **Fitted** or **Incremental** positioning method, and the total rotation of all occurrences is not 360°, pick the **Flip** button if necessary to pattern around the opposite rotation. Patterns form in a direction on one side or the other of the selected features, depending on the flip direction. An alternative is to pick the **Midplane** check box to spread the occurrences on each side of the selected items. You may still need to pick the **Flip** button if you enter an even number of total features. See **Figure 10-12**. Pick the **OK** button to pattern the features.

Figure 10-10.
Patterning a feature using the **Fitted** option.

Parent feature (hole)

90°

Selected axis
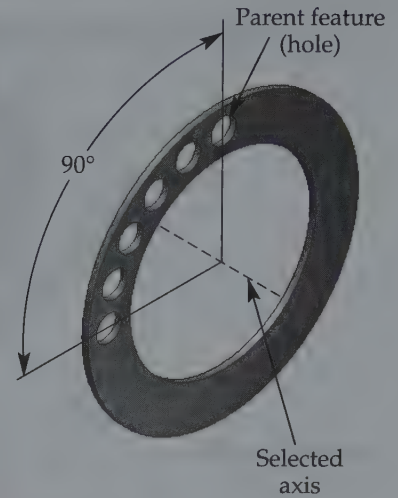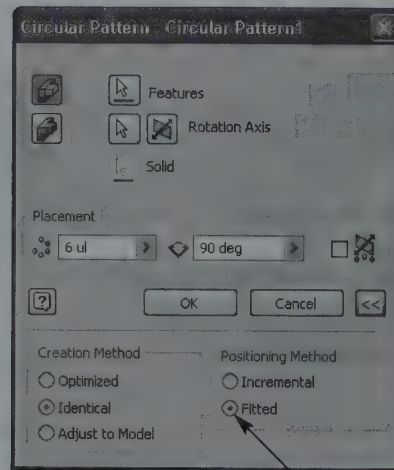
Pick to fit the pattern in the specified angle

Figure 10-11.
Patterning a feature using the **Incremental** option.

Parent feature (hole)

30°
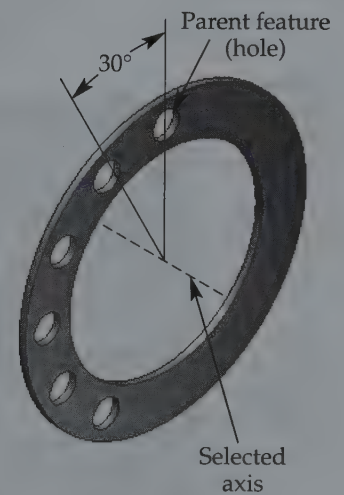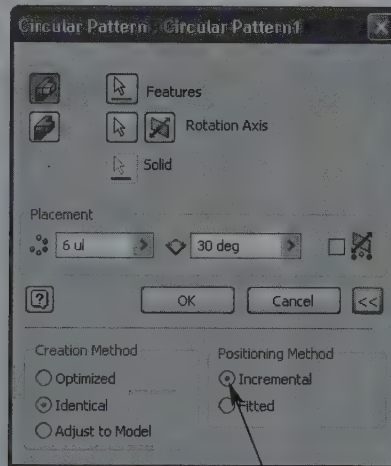
Selected axis

Pick to increment each occurrence at the specified angle

Figure 10-12.
Compare the locations of the patterned features when the **Midplane** direction option is selected and deselected. Select the **Flip** button to place the extra feature on the other side of the pattern.
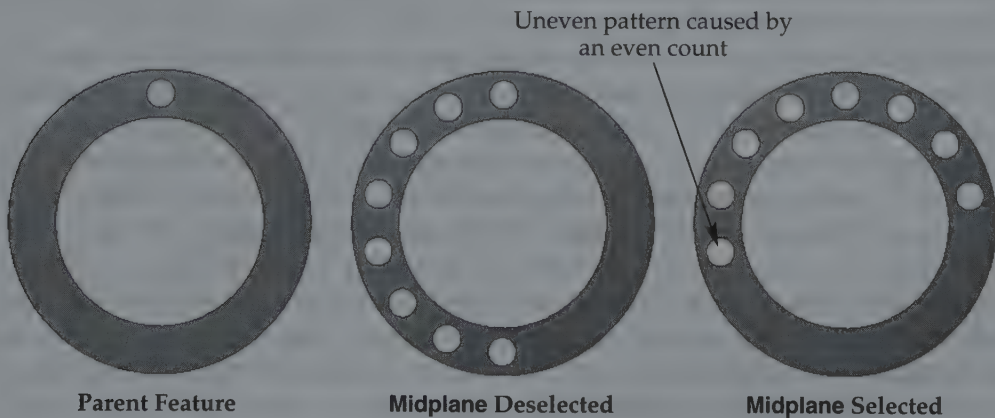
Uneven pattern caused by an even count

Parent Feature   **Midplane Deselected**   **Midplane Selected**

Editing Feature Patterns

You can edit and delete feature patterns and patterned items much as you would other part features. **Figure 10-13** shows an example of a part developed using each type of feature pattern. Study the corresponding information in the browser. Feature patterns appear in the browser much like other features, with an individual icon and default name. The first mirror operation in a model is Mirror1, the first rectangular pattern is Rectangular Pattern1, and the first circular pattern is Circular Pattern1.

Mirrored features and rectangular and circular patterns have the same browser characteristics. Expand the feature to display a **Features** folder. If you used the **Pattern individual features** option to create the pattern, the selected features, sketches, and other items consumed by the pattern appear in the folder, as shown in **Figure 10-13**. If you chose the **Pattern a solid** option, Pattern of Part, followed by any included work features and surfaces, appears in the **Features** folder. See **Figure 10-14**.

Following the **Features** folder is a list of each pattern occurrence. The selected parent features or entire solid display as the first, or top, occurrence. For mirrored features, only two occurrences are present: the parent selection and the mirrored occurrence. The specified count determines the number of rectangular or circular feature pattern occurrences. If you include work features, such as the axes or planes

Figure 10-13.
Compare the items in the browser to the related items in a model containing a circular feature pattern, mirrored feature, and rectangular pattern.
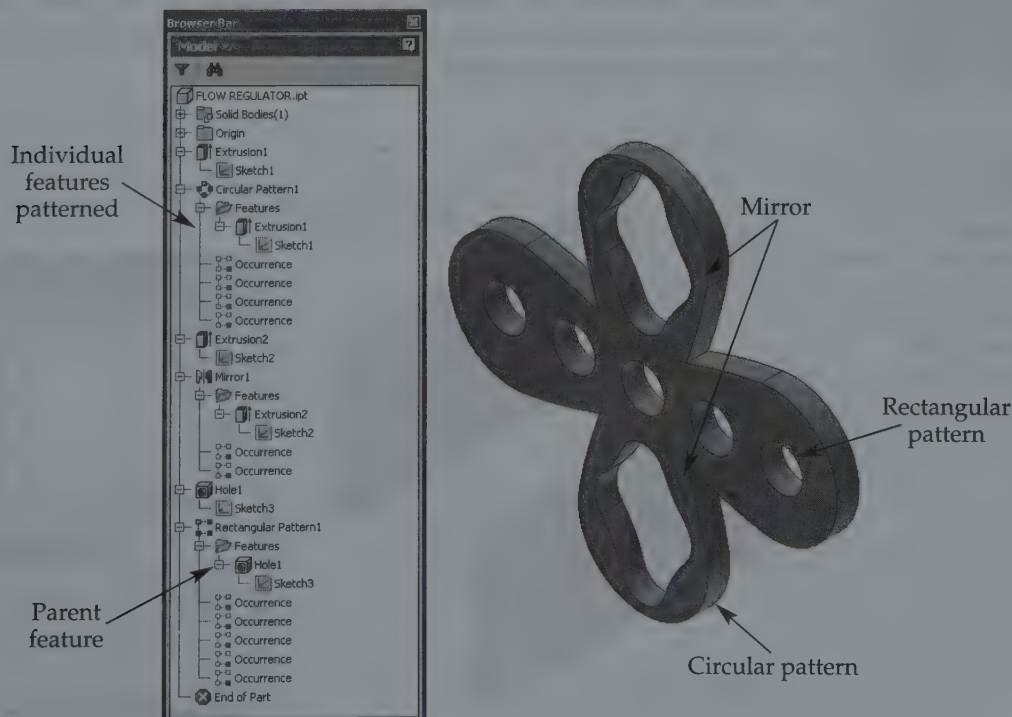
Figure 10-14.
A part formed by patterning all solids in the part, including two extrusions, a chamfer and a thickening, using the **Circular Pattern** tool. The browser groups all features into a **Pattern of Part** definition. The same occurs with mirrors and rectangular patterns.

Original features grouped for patterning

All solids patterned

Pattern occurrence

Circular pattern
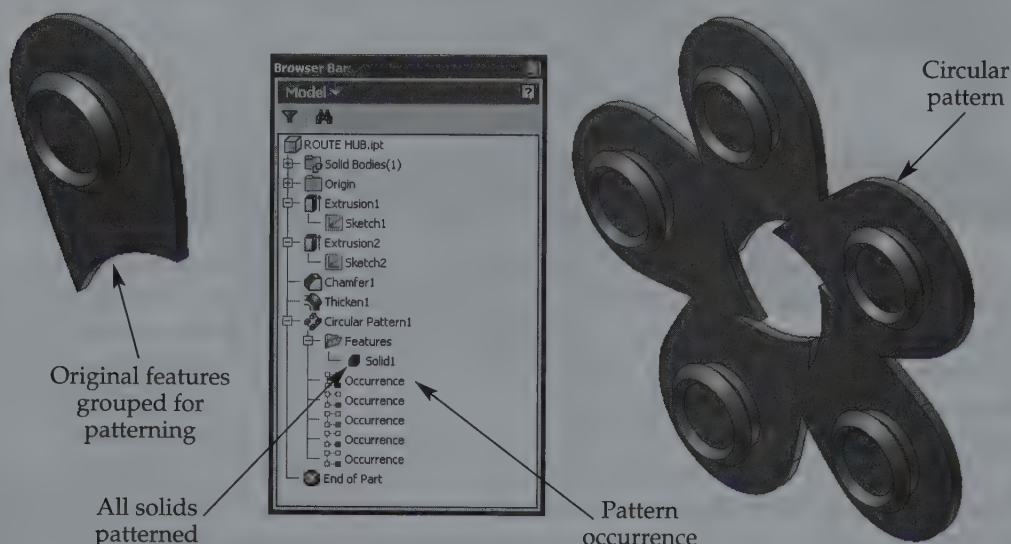
in the **Origin** folder of the browser, those features are grouped with each occurrence, except the parent occurrence. Expand an occurrence to view the selected features.

As with other items in the browser, when you move the cursor over the occurrence, the feature highlights in the graphics window. This is especially helpful when you are working with feature patterns to identify a specific occurrence in the model. To suppress, or "turn off," an occurrence, right-click on the occurrence and pick **Suppress**. You can suppress mirrored, rectangular, and circular pattern occurrences. **Figure 10-15** shows an example of suppressing a rectangular pattern occurrence.

> **NOTE**
>
> You cannot suppress the parent feature occurrence, but you can suppress the actual parent feature by right-clicking on the feature and selecting **Suppress Feature**.

Figure 10-15.
The effect of suppressing a rectangular pattern occurrence. You can also suppress mirrored and circular pattern occurrences.
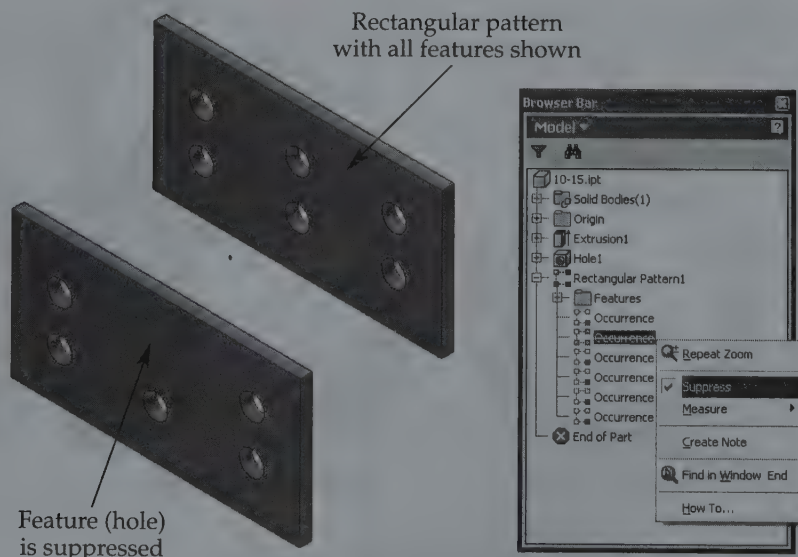
Rectangular pattern with all features shown

Feature (hole) is suppressed

Chapter Test

1. Describe feature patterns and explain when you might use them in developing a model.
2. What is the most efficient method of patterning multiple related features, such as an extrusion that contains a chamfer and a hole?
3. Describe a mirrored feature and give an example of its use.
4. What items can you choose as a mirror plane?
5. Describe rectangular feature patterns and give an example of their use.
6. If you want to pattern a feature that you cannot access directly in the model, how can you select the feature for patterning?
7. Briefly describe the difference between the **Spacing** option and the **Distance** option for creating a rectangular pattern.
8. Briefly describe circular feature patterns and give an example of their use.
9. Explain the difference in the browser items created using the **Pattern individual features** option and the **Pattern a solid** option of the feature patterning tools.
10. Explain the axis of rotation for a circular pattern and identify at least two items you can select as the axis.

Problems

1–2 Instructions:

- *Open the specified part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

1. **File:** P8-5.ipt

   Save as: P10-1.ipt

   Title: SUPPORT BRACKET

   Specific Instructions: Use the **Circular Pattern** tool to pattern the entire solid as shown. Select the default Z axis as the rotation axis, specify a count of 2, the **Incremental** position method, and an angle of 135°. The final part should look like the part shown.
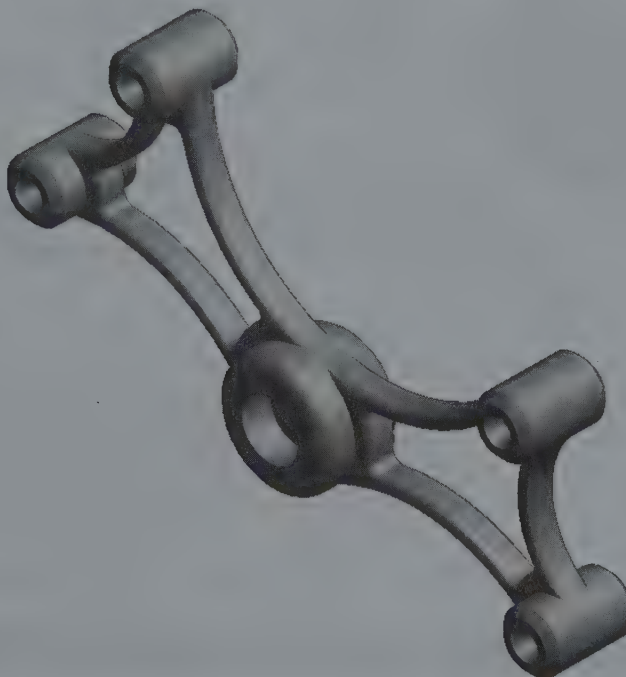
2. **File:** EX10-2.ipt
   Save as: P10-2.ipt
   Title: SELECTOR BRACKET
   Specific Instructions:
   A. Sketch the Ø27.5 mm circles shown on the top face, and extrude them 10 mm.
   B. Suppress six of the rectangular pattern holes, as shown.
   C. Use the **Fillet** tool to fillet and round All Fillets and All Rounds by 2 mm.
   D. Add the specified holes.
   E. Open the **iProperties** dialog box for the model and enter SELECTOR BRACKET in the **Title** text box, and enter P10-2 in the **Part Number** text box.
   F. Choose an appropriate material and color.
   The final part should look like the part shown.

Ø5 THRU
⌴ Ø10 ⬇ 3

3–5 Instructions:

- *Create sketches of the following objects*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

3. **Title:** HUB
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-015-01
   Project: HUB
   Material: Default
   Color: Default
   Save as: P10-3.ipt
   Specific Instructions: Sketch a ∅1.5″ circle on the XZ plane. Extrude the sketch 5″ in the positive direction. Add the specified hole as shown.

1-8 UNC - 2B
⌴ ∅1.25 ▼ 2.50

4. **Title:** SUPPORT
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-016-01
   Project: LATHE
   Material: Default
   Color: Default
   Save as: P10-4.ipt
   Specific Instructions:
   A. Create the sketch shown on the XY plane and extrude the sketch 2″ midplane.
   B. Create the sketch shown on the top face and cut-extrude the sketch 1″ in the negative direction, as shown.
   C. Mirror the cut extrusion using the default XY plane as the **Mirror Plane**.
   D. Add the specified hole .5″ from the bottom edge and 1″ from the vertical edge, as shown.
   E. Place a .028625″ chamfer at the hole edges.
   F. Place a .0625″ round on **Extrusion1**.

   The final part should look like the part shown.

Projected center point

1/2-13 UNC – 2B
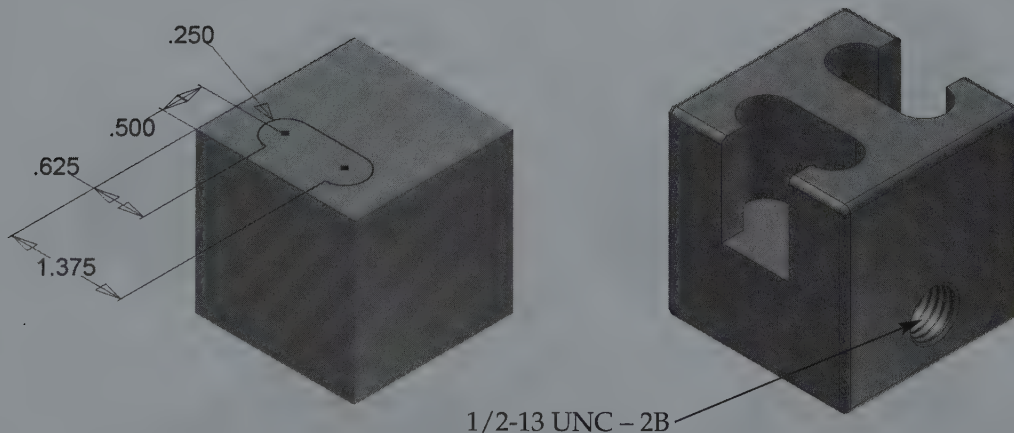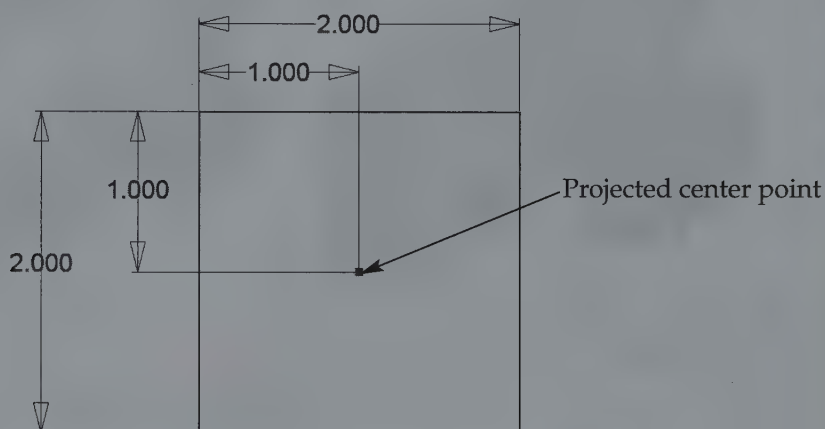
5. **Title:** FLOW RESTRICTOR
   Units: Metric or Inch
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-017-01
   Project: FLOW RESTRICTOR
   Material: Choose a material
   Color: Choose a color
   Save as: P10-5.ipt
   Specific Instructions: Create features similar to those shown, using your own specifications. Mirror the features twice to create the part shown. Place rounds on the two loops shown.

Rounds

Second mirror plane
(not visible)

First mirror plane
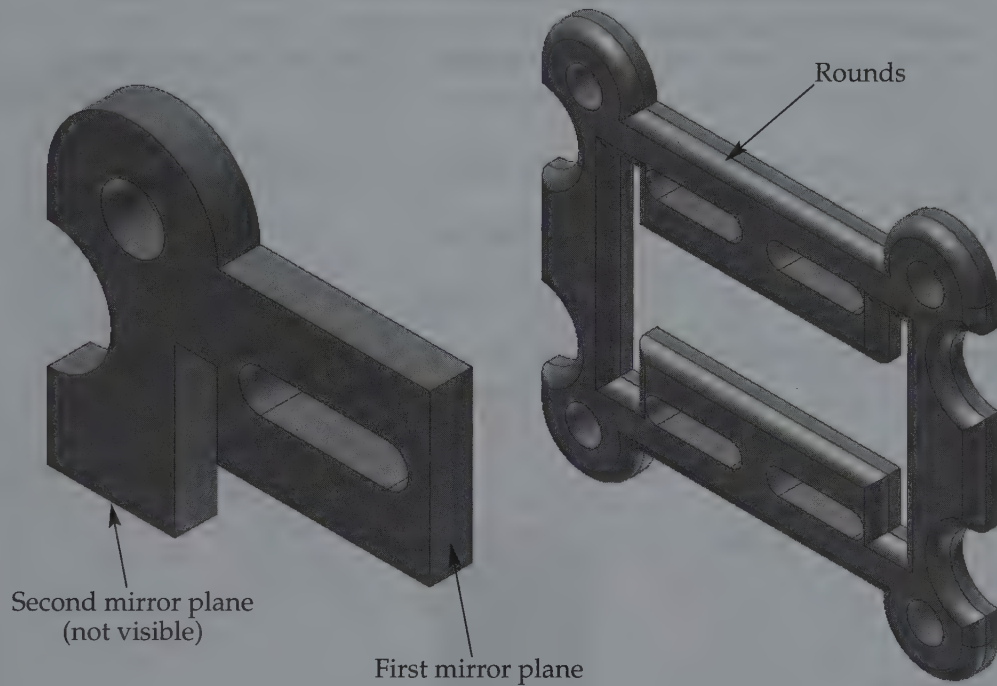
6. **Title:** SLIDER HINGE CONNECTOR
   Units: Metric
   Template: Part-mm.ipt
   Part Number: IAA-018-01
   Project: SLIDER HINGE
   Material: Titanium
   Color: As Material
   Save as: P10-6.ipt
   Specific Instructions:
   A. Create the sketch shown on the XY plane and extrude the sketch 100 mm in the positive direction, as shown in A.
   B. Place a 5 mm fillet as shown in B.
   C. Create the sketch shown in B on the specified face and cut-extrude the sketch 10 mm in the negative direction.
   D. Use the **Rectangular Pattern** tool to pattern the extrusion as shown in C. Use a count of 5 and a spacing of 22.5.
   E. Add a 10 mm hole through all the feature geometry as shown in C.
   F. Place a 10 mm fillet on the edges as shown in C.
   G. Mirror the entire solid as shown in D.
   H. Add the specified hole, coincident to the edge shown.
   I. Use the Rectangular Pattern tool to pattern the hole as shown. Use a count of 5 and a spacing of 13.75.
   The final part should look like the part shown.

Projected center point

A

12.5

5 mm fillet

B

10 mm fillet

C

Coincident

Ø5 THRU
⊔ Ø10 ▽ 3

D
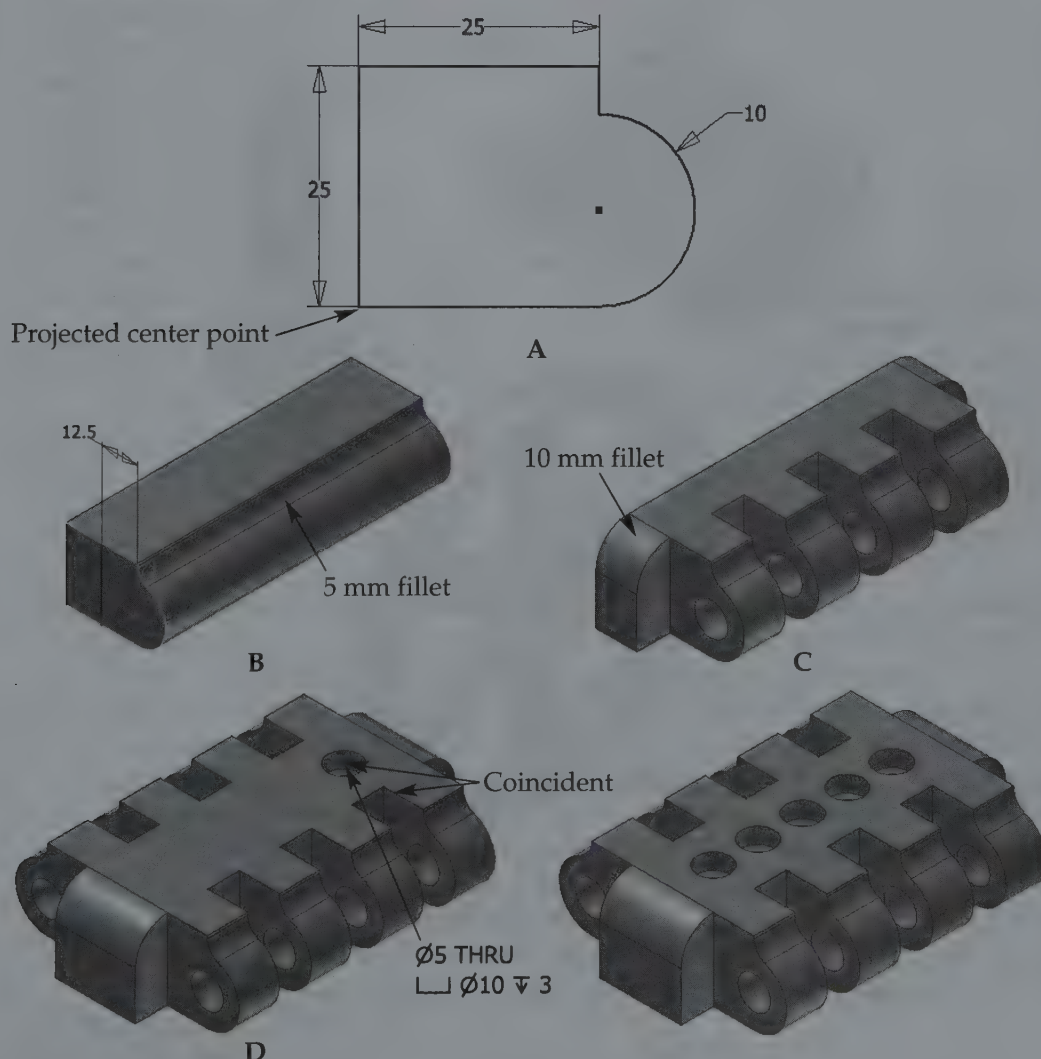
7. **Title:** WHEEL
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-019-01
   Project: GATE VALVE
   Material: Steel, Mild
   Color: Orange (Dark)
   Save as: P10-7.ipt
   Specific Instructions:
   A. Sketch a ∅2″ circle around the projected center point, on the XY plane, and extrude the sketch 2″ midplane.
   B. Sketch the geometry shown on the XZ plane as shown in A. **Slice Graphics** to clarify the sketch environment.
   C. Extrude the sketch 6″ in the positive direction.
   D. Sketch the circle shown in B on the YZ plane. Fully revolve the sketch around the Z axis.
   E. Use the **Circular Pattern** tool to pattern Extrusion2 as shown. Specify the default Z axis as the rotation axis, a count of 6, a fitted position method, and 360° angle.
   F. Add the specified hole to Extrusion 1.
   The final project should look like the part shown.

Align

.500

1.000

1.500

.375

Projected center point

A

B

1-20 UNEF - 2B

C
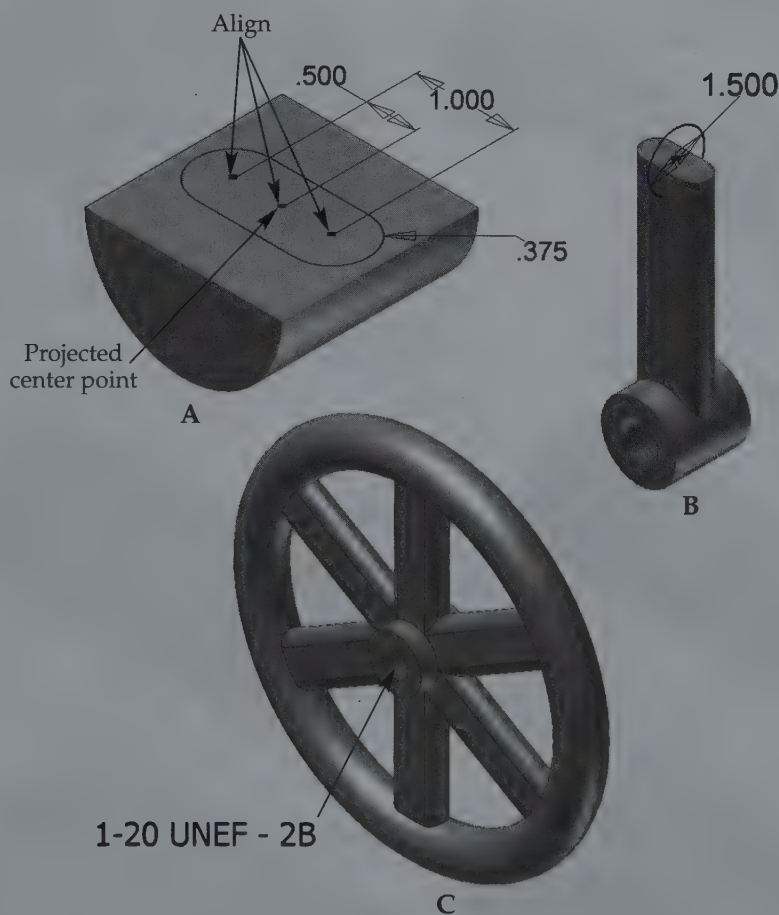
8. **Title:** BOTTLE CAP
   Units: Metric
   Template: Part-mm.ipt
   Part number: IAA-014-02
   Project: BOTTLE
   Material: Choose a material
   Color: Choose a color
   Save as: P10-8.ipt
   Specific Instructions: Develop a cap for the bottle you created in Problem 9-5. Use the **Circular Pattern** tool to apply straight knurls to the cap as shown. Sketch the parent knurl feature from one of the reference planes located in the **Origin** folder, and extrude through the cap. Then apply the circular pattern, followed by the hole.
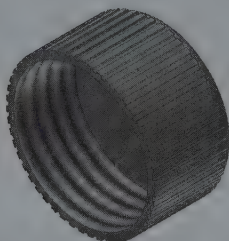
9. **Title:** 45° ELBOW
   Units: Metric
   Template: Part-mm.ipt
   Part number: IAA-020-01
   Project: PIPING
   Material: Steel, Mild
   Color: As Material
   Save as: P10-9.ipt
   Specific Instructions:
   A. Create the sketch shown in A on the XZ plane.
   B. Revolve the profile formed by both circles around the Z axis 45° in the positive direction, as shown in B.
   C. Sketch a Ø108 mm circle on the bottom face of the revolved feature, as shown in C. Project the inside of the revolution to include in the sketch. The Ø108 mm circle should be concentric to the projected edge.
   D. Extrude the donut-shaped sketch profile 11.5 mm away from the revolution, as shown in D.
   E. Add a Ø16.66 mm hole, as shown in E. The hole should be horizontal to, and 41.5 mm from, the center of the previous extrusion.
   F. Use the **Circular Pattern** tool to pattern the hole, as shown in F. Select the curved Extrusion1 (likely default name) face (cylinder wall) as the rotation axis. Specify a count of 8, the fitted position method, and a 360° angle.
   G. Copy Extrusion1 to the top face of the revolved feature by dragging Extrusion1 from the browser into the graphics window. Refer to G and use the **Paste Features** dialog box to paste the extrusion and the pattern of holes: Choose **Dependent** from the paste features drop-down list to include the pattern of holes. Select the top face of the revolution as shown to position Profile Plane1 as identified in the **Name** list. Pick the **Finish** button to create the copy shown in H.
   H. The pasted extrusion remains unconstrained. Edit the extrusion sketch to add coincident constraints linking the sketches to the projected revolution edge, and a tangent constraint associating the small circle with the corresponding projected revolution edge.
   I. The final project should look like the part shown in J.

108

3.9

57.8

Projected   Horizontal
center point
A

B

108
C

D

E

F

10.8
G

H

108
I

J

11
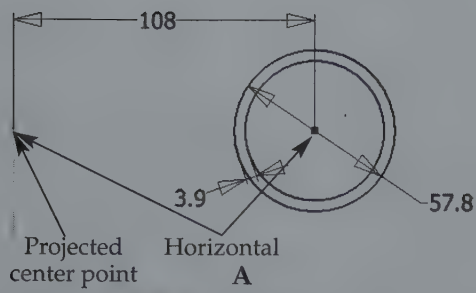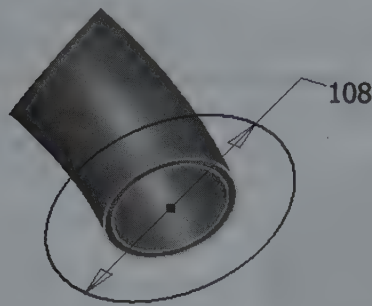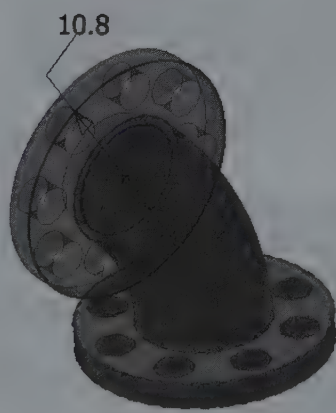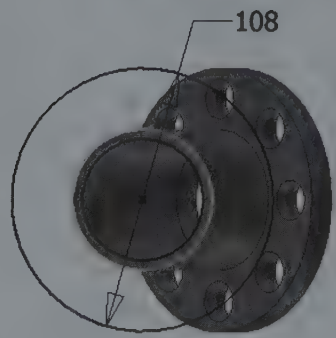
Work Features

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create work planes.
✓ Project cut edges.
✓ Establish work axes.
✓ Use ungrounded and grounded work points.
✓ Establish a user coordinate system (UCS).
✓ Adjust work features and the UCS.

Often while developing models, you will not find sufficient geometry on which to sketch, constrain, and build features. Use *work features* to solve many of these problems and to create complex shapes. Work features provide the same functions as the default work planes, axes, and center point found in the **Origin** folder of the browser, but they offer the flexibility to reference planes, axes, or points anywhere needed.

work features: Construction points, lines, and surfaces that create reference elements anywhere in space to help position and generate additional features.

Work Planes

Add a *work plane* whenever you require a planar construction surface and a default reference plane or feature face is unavailable, such as positioning a sketch, locating a work feature, adding a plane to terminate a feature, or creating a constraint plane for an assembly. **Figure 11-1** shows an example of using a work plane to position an extrusion sketch. A work plane is infinite, though you can only see a portion of it in the graphics window, and it appears semitransparent by default.

work planes: Flat reference surfaces located anywhere in space.

Access the **Work Plane** tool to create a work plane. You can use a variety of methods to place a work plane, depending on the application. Common techniques include offsetting a work plane from an existing face or plane, or picking feature or sketch geometry. Options are available for adjusting work plane appearance and definition after it has been placed.

Ribbon

Model
> Work Features

Work Plane

Type

[ ]

Figure 11-1.
A—Offsetting a work plane from an existing face. B—Sketching on and extruding from the work plane (sketch visible for reference). C—Adding features to complete the part.

Offset from face

.500
fx:.250

fx:.375
.750

A   B   C
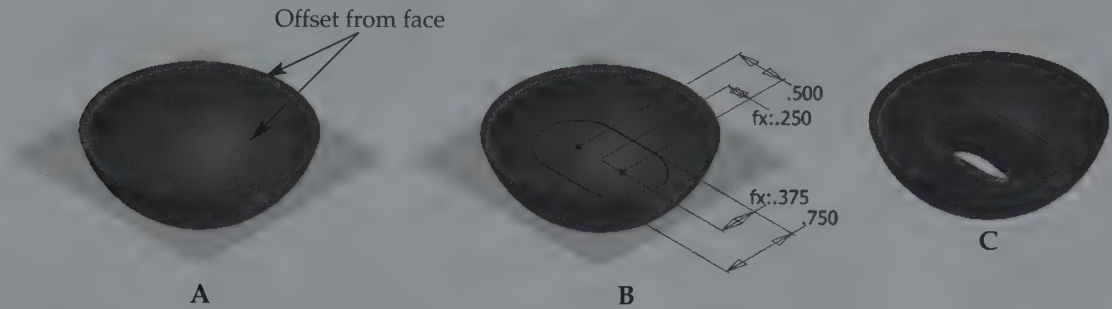
Offsetting a Work Plane

One of the most common techniques for positioning a work plane is to offset it from a plane or planar face. Select a plane or face to display a work plane preview and icon. To offset the work plane, hold down the left mouse button on the plane preview and drag in the appropriate direction. The **Offset** dialog box appears for precise placement. See **Figure 11-2**. Release the mouse button and specify a value using the text box.

If you plan to begin a new 2D sketch on an offset work plane, use the **2D Sketch** tool without first placing a work plane. Instead of picking a plane or face, as you would to begin a new sketch on the plane or face, hold down the left mouse button on the plane or face and drag in the appropriate direction. Use the **Offset** dialog box to specify the offset.

Ribbon
Model
> Sketch

2D Sketch

Shortcut
New Sketch
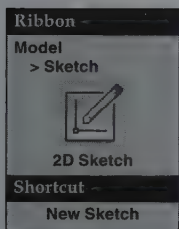
Exercise 11-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-1.

Figure 11-2.
An example of offsetting a work plane from the default XY plane and from a feature face.

Offset work plane

Origin plane
(XY plane)

Feature
face

Specify offset
distance

Offset
0.375 in

Offset
0.500 in

Specify offset
distance

Offset
work plane

Offset from Origin Plane   **Offset from Feature Face**
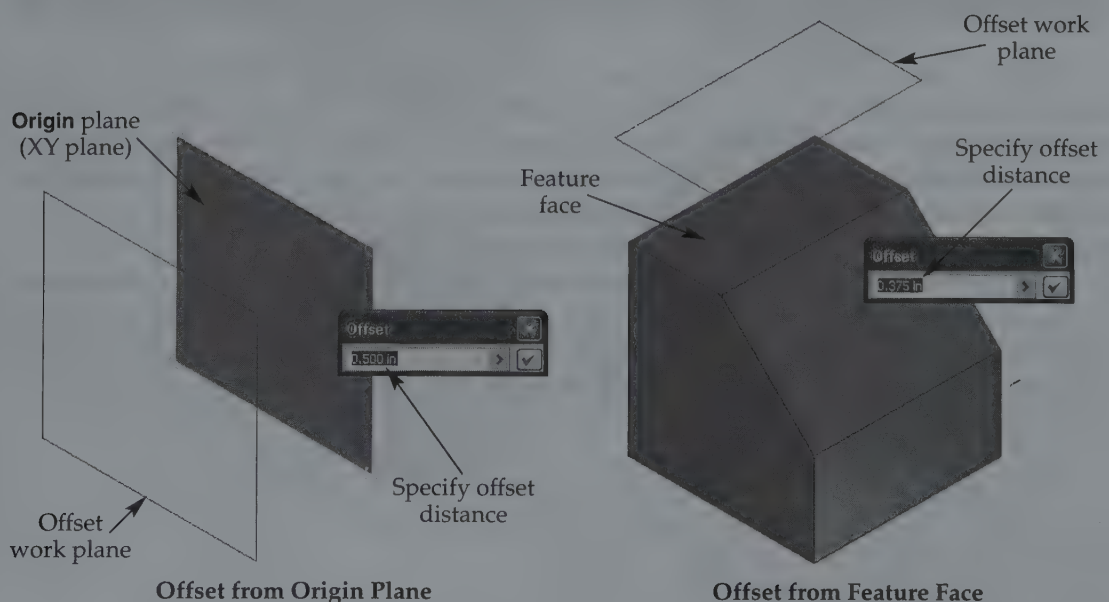
Picking Feature or Sketch Geometry

Several options are available for placing a work plane when it is not appropriate or possible to offset from a plane or face. **Figure 11-3** shows picking two parallel planes or planar faces to position a work plane parallel to and bisecting the distance between the planes or faces. **Figure 11-4** shows picking two edges or axes to position a work plane *coplanar* to the edges or axes. **Figure 11-5** shows picking three points to create a work plane coincident to the points. Selectable points include feature edge endpoints, intersections, midpoints, and work points.

coplanar: Lying in the same 2D plane.

Figure 11-6A shows picking an edge or axis and a point to create a work plane perpendicular to the edge or axis, through the point. **Figure 11-6B** shows picking a face or plane and a point to position a work plane parallel to the face or plane, through the point. Pick a face or plane and an edge or axis to locate a work plane at an angle to the face or plane, through the edge or axis. The edge acts as the angle pivot point. As shown in **Figure 11-7**, after you pick an edge or axis and a face or plane, the **Angle** dialog box appears, allowing you to specify the angle of the work plane. You can select any face or plane parallel to the selected edge and create the same work plane by entering different angles.

Figure 11-3.
Creating a work plane that bisects two existing planes or planar faces.

Existing work planes used to create Work Plane1

Existing face (not shown) used to create Work Plane2

Work Plane1

Existing face used to create Work Plane2
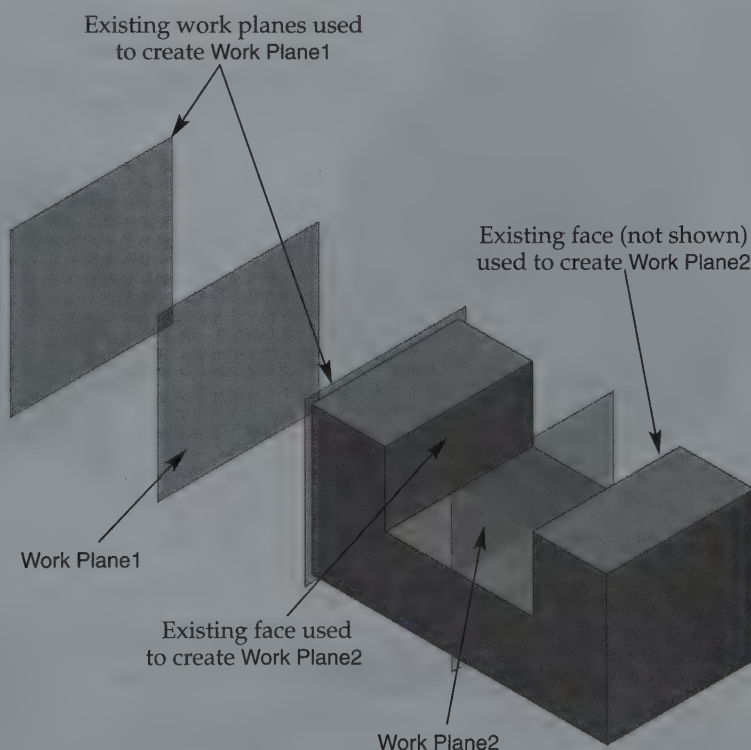
Work Plane2

Figure 11-4.
Positioning a work plane coplanar to two axes or edges.

Axes
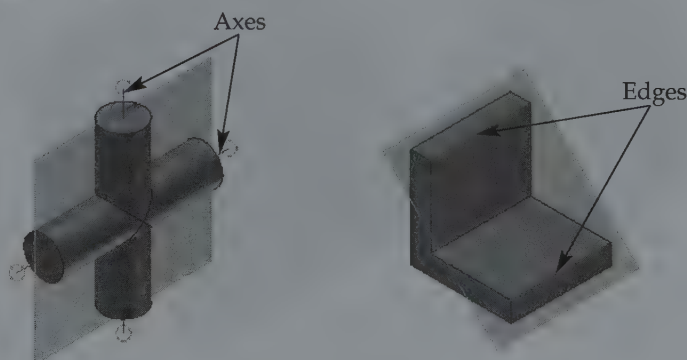
Edges

Figure 11-5.
Locating a work
plane coincident to
three points.
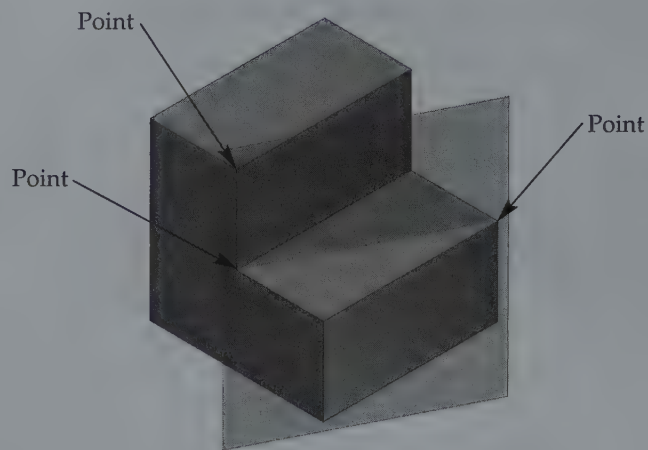
Point

Point

Point

Figure 11-6.
A—Creating a work plane by selecting an edge and point. B—Positioning a work plane by
selecting a face and point.

Point
(center point
of arc)

Point
(feature edge
midpoint)

Edge

Face

A

B
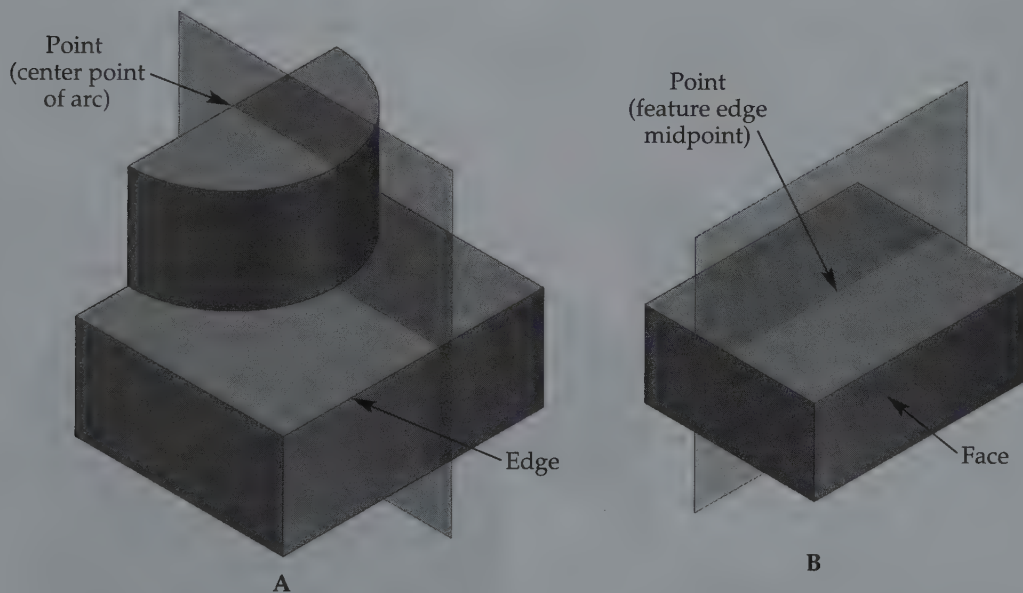
Figure 11-7.
Examples of work
planes at edges, at
specified angles
from faces.

Angle

135

Angle

90

Face (135° or –45° angle)

Edge

Edge

Face (90° or 270° angle)
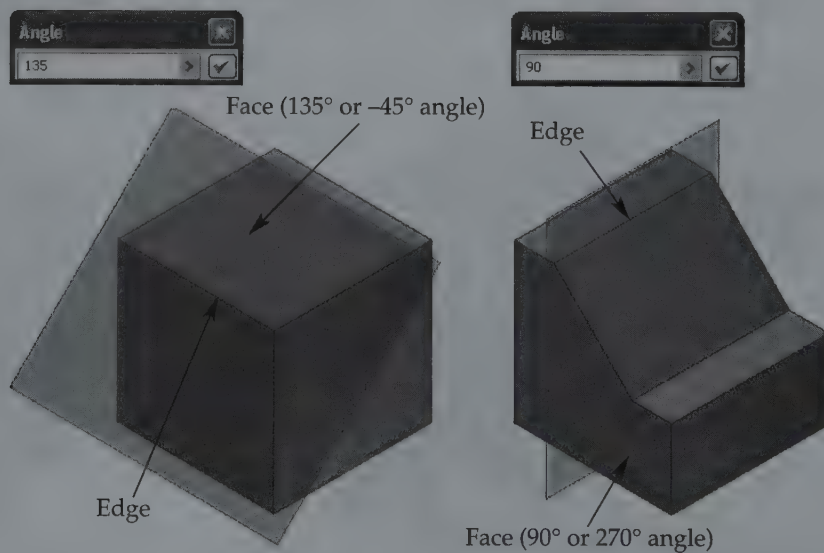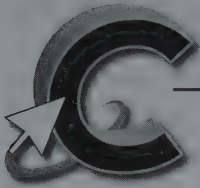
Exercise 11-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-2.

Pick a sketch curve and a point on the curve to generate a work plane perpendicular to the curve plane, at the point. This is a common technique when constructing a sweep profile and path, as described in Chapter 12, and is also necessary for other applications. **Figure 11-8** shows examples of using a sketch to position a work plane.

In order to create a work plane tangent to a curved surface, such as an extruded cylinder, another feature or sketch must be available for selection. If you sketch the curved feature geometry at the projected center point and want the work plane to be parallel to an origin work plane, select the cylinder surface and the appropriate origin work plane. See **Figure 11-9A**. If the work plane should not be parallel to an origin work plane, select the appropriate origin work plane and axis, and use the **Angle** dialog box to specify the angle of the work plane. Then create the final work plane by selecting the cylinder surface and the angled work plane. See **Figure 11-9B**.

Figure 11-8.
A—Creating a work plane perpendicular to a curve sketched on the XZ plane, at a point on the curve. B—Forming a work plane perpendicular to a curve sketched on a feature face, at a point on the curve.
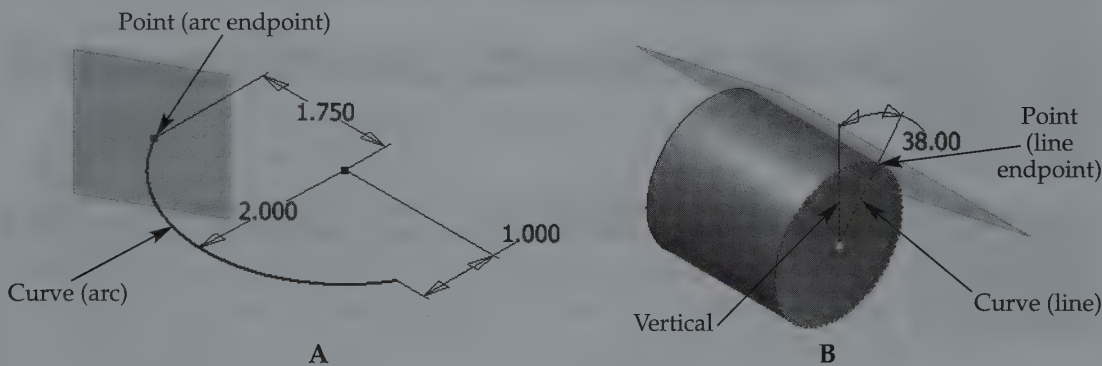
Figure 11-9.
A—Selecting an origin work plane and cylinder surface to create a tangent work plane.
B—Follow these steps to create a tangent work plane at an angle from an origin work plane.
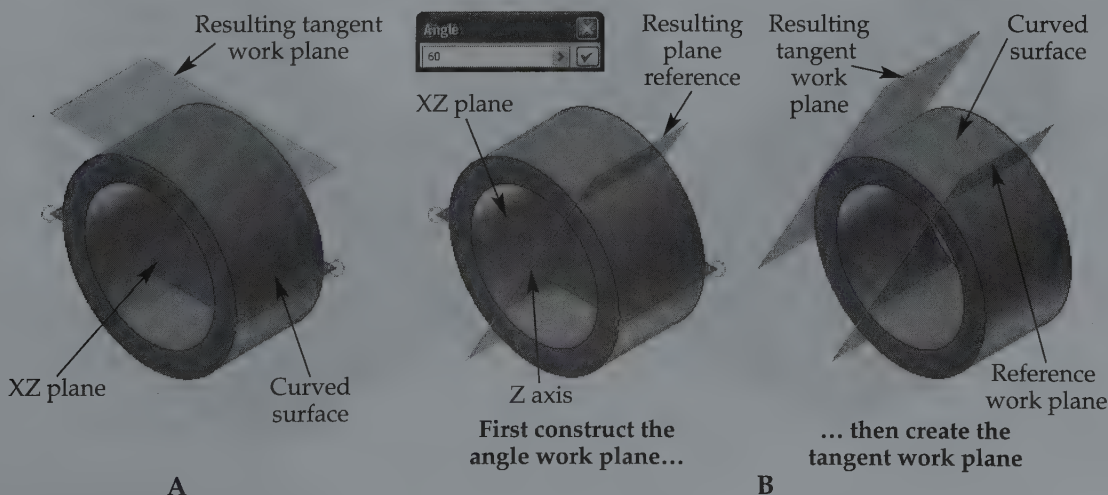
To create a work plane tangent to a curved surface when you cannot reference origin work features, pick a sketched curve and point as shown in **Figure 11-9B**. The work plane becomes associated with the sketch. In order for the work plane to be tangent to the curved surface, the sketch plane must be perpendicular to the curved surface, the curve must be coincident to the axis of the curved surface, and the point must be coincident to the tangent edge.

Exercise 11-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-3.

Figure 11-10A shows picking a curved surface and an edge or axis to create a work plane tangent to the curved surface, coplanar to the edge or axis. **Figure 11-10B** shows picking a curved surface and a face or plane to define a work plane tangent to the curved surface, parallel to the face or plane.

Figure 11-10.
A—Creating a work plane by selecting a tangent face and an edge. B—Adding a work plane by selecting a tangent face and a planar face.

Tangent face

Edge

A

Tangent face

Planar face
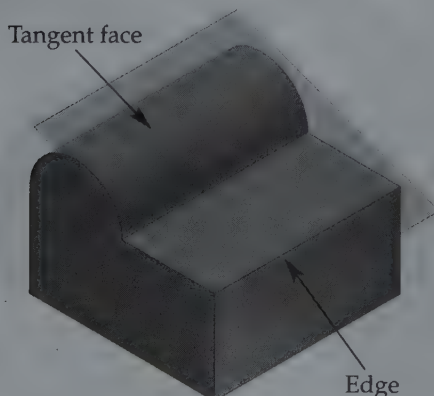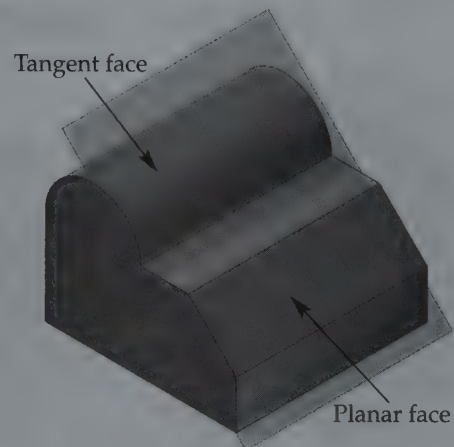
B

Exercise 11-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-4.

Projecting Cut Edges

The **Project Geometry** tool is effective for common applications that require projecting content onto a sketch plane, such as the center point or feature edges. The **Project Cut Edges** tool allows you to project content that you cannot reference using the **Project Geometry** tool, most often geometry at the intersection of a sketch plane positioned on a work plane. See **Figure 11-11**. When you access the **Project Cut Edges** tool in the part environment, all feature geometry cut by the sketch plane automatically projects onto the sketch as edges; there is no need to pick the solid or the intersection individually.

Ribbon
Sketch
> Draw

Project Cut Edges

> **NOTE**
>
> In the assembly environment, to associate cut edges from a component onto a sketch plane of another component, you must pick the component to project.

Exercise 11-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-5.

Figure 11-11.
The **Project Cut Edges** tool allows you to use geometry created by the intersection of a face or plane and feature geometry that would be difficult or impossible to reference otherwise.
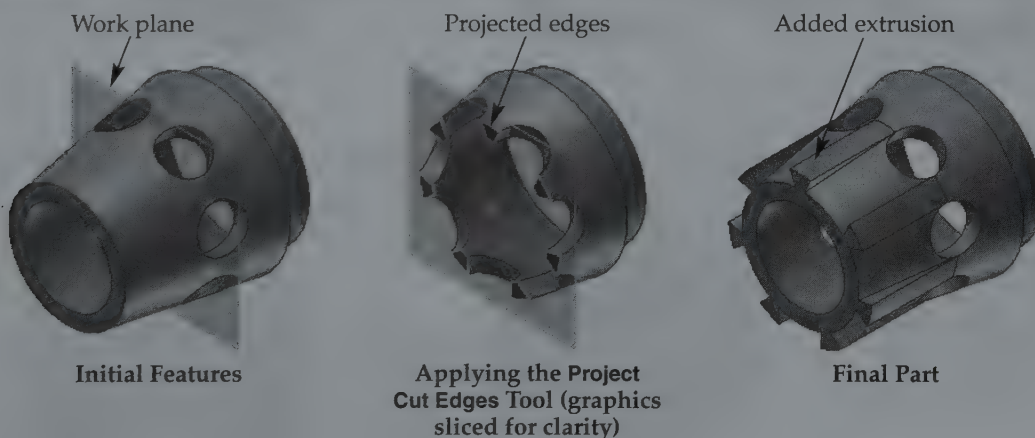
Work plane   Projected edges   Added extrusion

Initial Features   Applying the Project   Final Part
                        Cut Edges Tool (graphics
                        sliced for clarity)
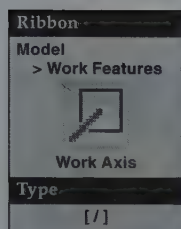
Add a *work axis* when you require a construction line or axis and a default reference axis or feature axis or edge is unavailable, such as when creating a circular feature, locating a work feature, or creating a constraint axis or edge for an assembly. **Figure 11-12** shows an example of using a work axis to revolve a sketch profile. The function of work axes is sometimes limited due to the availability of origin axes and the ability to select a sketch line or cylindrical feature when creating circular features. However, you will find applications when a work axis is necessary, or easier to use than an alternative, such as a sketch line. A work axis is infinite, although you can only see a portion of it in the graphics window, and appears as a thin line.

Access the **Work Axis** tool to create a work axis. Select feature or sketch geometry to place a work axis, depending on the application. Options are available for adjusting work axis appearance and definition after it has been placed.

Ribbon

Model
> Work Features

Work Axis

Type

[/]

Picking Feature Geometry

You can reference a variety of feature geometry to place a work axis. **Figure 11-13A** shows picking a curved feature to add the work axis through the center. **Figure 11-13B** shows selecting a linear edge to locate a work axis collinear to the edge. **Figure 11-14A** shows picking two points to create a work axis coincident to the points. **Figure 11-14B** shows selecting two faces or planes to locate a work axis coplanar to and at the intersection of the faces or planes. **Figure 11-15A** shows picking a face or plane

Figure 11-12.
A—Positioning a work axis between existing faces. B—Using the work axis to revolve a sketch profile. C—Adding an extrusion to complete the part.

Bisects surface

A

B

C
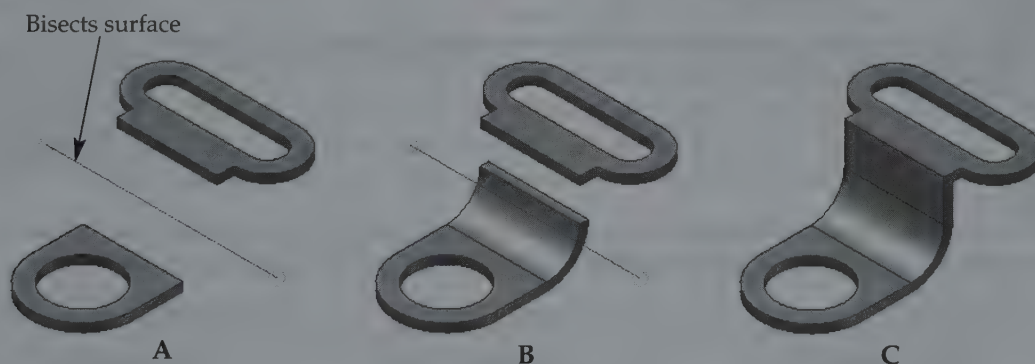
Figure 11-13.
A—Creating a work axis by picking a revolved or cylindrical feature. B—Adding a work axis collinear to a linear edge.

Work axis

Selected edge

Work axis

Selected feature

A

B
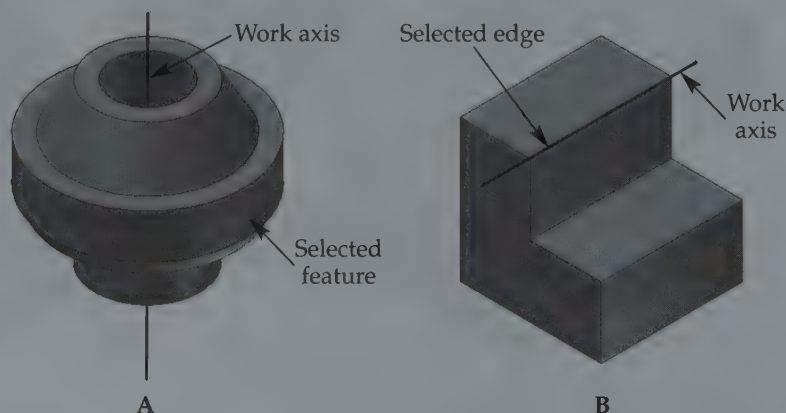
Figure 11-14.
A—Placing a work axis by selecting two points. B—Creating a work axis at the intersection of two work planes. You can select faces or a combination of faces and planes.

Point (endpoint)

Work plane

Work plane

Work axis

Point (endpoint)

A

B
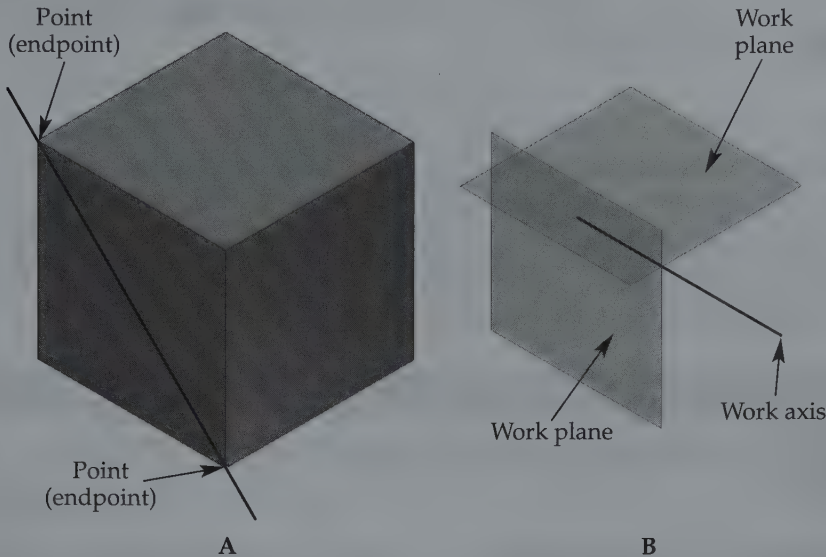
Figure 11-15.
A—Placing a work axis by selecting a face or plane and a point. B—Creating a work axis by selecting a face and an edge.

Selected point (edge midpoint)

Work axis

Selected face

Selected face

Selected edge

A

B
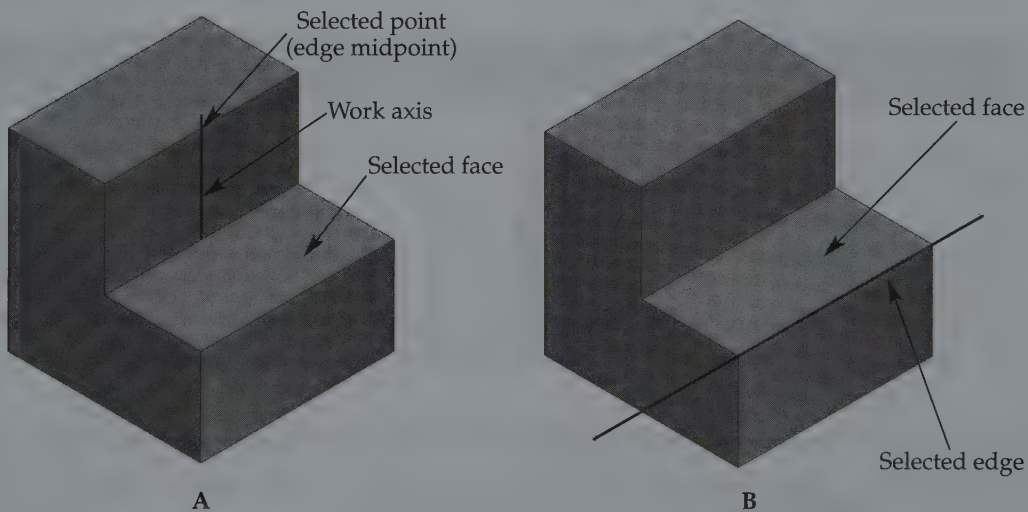
and a point to place a work axis perpendicular to the face or plane, through the point. **Figure 11-15B** shows selecting a face or plane and an edge to create a work axis at the edge of the face and parallel to the edge.

Exercise 11-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-6.

Exercise 11-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-7.

Using Sketch Geometry

Existing 2D and 3D sketch geometry offers multiple options to locate work axes. For example, select a linear 2D or 3D sketch line, or pick sketch points such as endpoints, midpoints, and center points. Another option is to select a sketch point in combination with any existing work point, feature face, plane, or axis.

> **NOTE**
>
> You can select items in any order to establish a work axis.
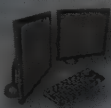
> **PROFESSIONAL TIP**
>
> Use an origin work axis instead of creating a work axis whenever possible, unless you want the axis to adapt to feature changes.

Work Points

work points:
Parametric reference points located on any part feature or in 3D space.

Ribbon

Model
> Work Features

Work Point

Type

[·]
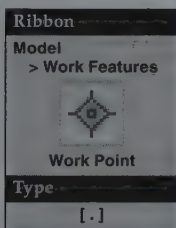
Work points are commonly used to create 3D sketch geometry, locate a work feature, terminate features, and establish constraint points for assemblies. **Figure 11-16** shows an example of using a work point to help form a loft. Chapter 12 explains loft features. The function of work points is sometimes limited due to the ability to select the center point and points on features. However, you will find applications when a work point is necessary, or easier to use than an alternative, such as a feature point. A work point appears as a highlighted work point icon in the graphics window.

Figure 11-16.
A—Positioning a work point at the midpoint of an edge. B—Lofting from a sketch profile to the work point. C—Adding features to complete the part.
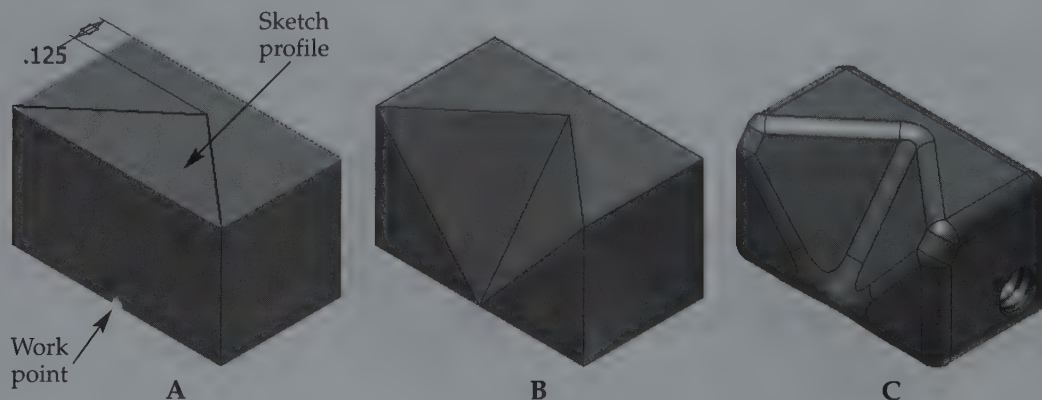
Access the **Work Point** tool to create a work point on selected feature or sketch geometry. A **Grounded Work Point** tool is also available for adding a work point in 3D space and is described later in this chapter. Options are available for adjusting work point appearance and definition after it has been placed.

Picking Feature Geometry

You can reference a variety of feature geometry to place a work point. **Figure 11-17** shows picking feature edge endpoints or a midpoint to place a work point. **Figure 11-18** shows selecting three planes or faces to place a work point at the intersection. **Figure 11-19A** shows picking a face, plane, or surface and an edge or axis to place a work point at the intersection. **Figure 11-19B** shows picking two edges or axes to position a work point at the intersection, or at the projected intersection if the selections do not meet.

Exercise 11-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-8.

Exercise 11-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-9.

Figure 11-17.
Visible work points created by selecting single points.

Selected points
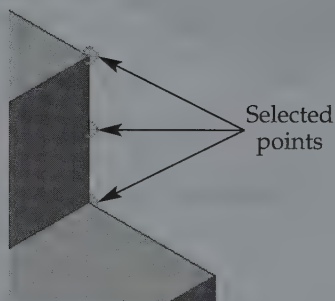
Figure 11-18.
Creating a work point at the intersection of three work planes or feature faces.

Selected faces

Selected planes

Work point
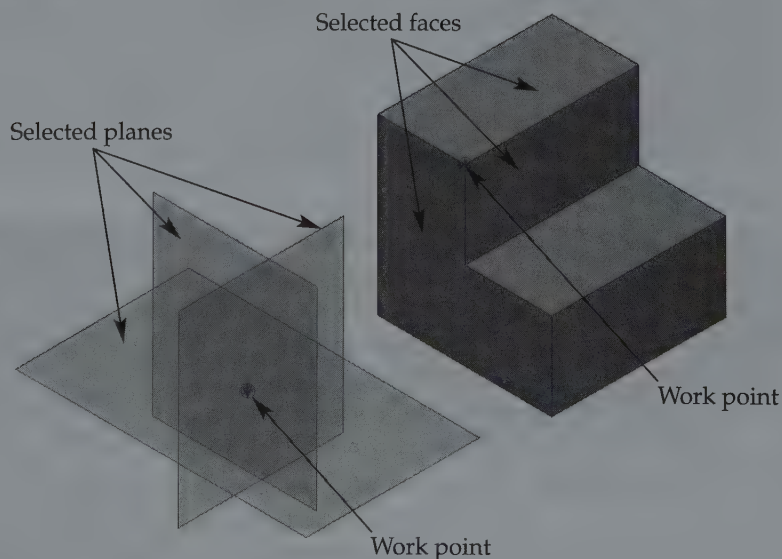
Work point

Figure 11-19.
A—Placing a work point by selecting a face and edge or axis. B—Adding a work point at the intersection or apparent intersection of two selected axes or edges.

Selected axis

Work point

Selected face

Selected face

Work point

Selected edge

A

Selected edge

Work point

Selected axis

Work point

Selected axis

Selected edge

Selected edge

Work point

Selected edge

Intersection

Apparent Intersection

B
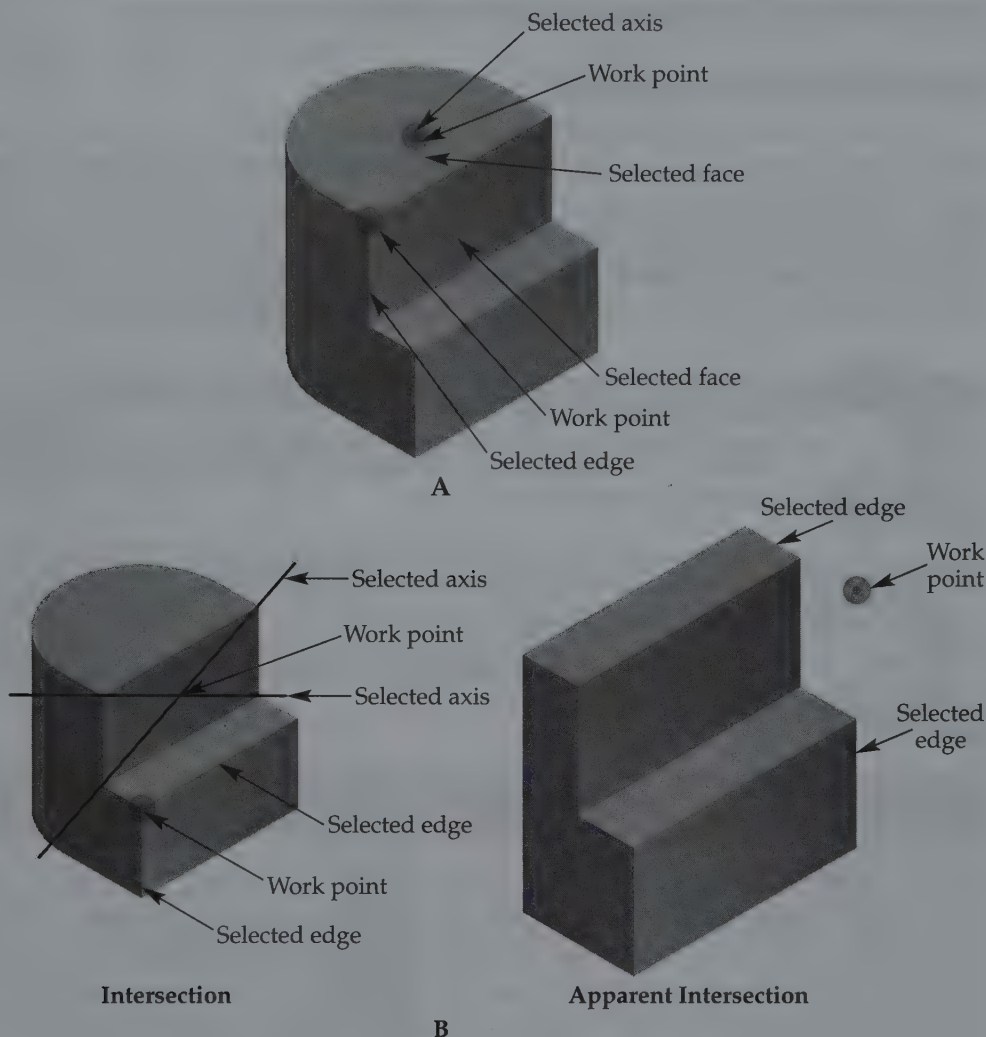
Using Sketch Geometry

Existing 2D and 3D sketch geometry offers a variety of options to locate work points. For example, you can define a work point by selecting two intersecting, linear 2D or 3D sketch lines or by picking sketch points such as endpoints, midpoints, and center points. Another option is to select a sketch curve such as a circle, arc, spline, or ellipse in combination with any existing face or plane.

Grounded Work Points

grounded work point: A work point completely fixed to an XYZ coordinate.

A *grounded work point* is located in 3D space, but is not fixed to model geometry. Therefore, grounded work points do not respond to model constraint changes. For example, if you place a grounded work point at the end of a 5″ line and then change the length of the line to 6″, the grounded work point remains at the initial XYZ location—in this example, the 5″ line endpoint.

Ribbon

Model
> Work Features

Grounded Work
Point

Type

[;]

Access the **Grounded Work Point** tool to create a grounded work point. Select a point on an existing feature to specify the initial location of the grounded work point. The **3D Move/Rotate** dialog box and triad appears as shown in **Figure 11-20**. The triad includes a sphere, axes, planes, and arrowheads that allow you to adjust the location of the work point. **Figure 11-21** shows the appearance of the **3D Move/Rotate** dialog box
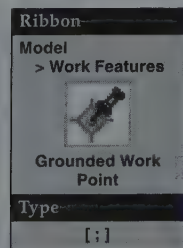
Figure 11-20.
A grounded work
point triad and the
3D Move/Rotate
dialog box.
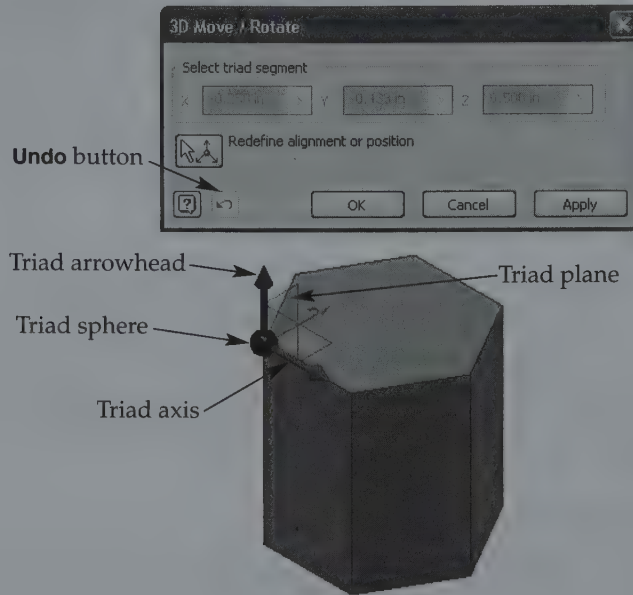
when you select the triad sphere or a plane, arrowhead, or axis. Refer to this figure as
you read the following descriptions of the grounded work point operations.

Unrestricted Move

The **Unrestricted move** option, available when you select the triad sphere, allows
you to move the grounded work point anywhere in 3D space. Pick the sphere and
specify values using the **X**, **Y**, and **Z** text boxes in the **3D Move/Rotate** dialog box. You
can also hold down the left mouse button on the sphere and drag the point.

Planar Move

The **Planar move** option, available when you pick a triad plane, allows you to move
the grounded work point anywhere on the selected plane. Pick a plane and specify
values using the corresponding text boxes in the **3D Move/Rotate** dialog box. For
example, the **X** and **Y** text boxes are available when you pick the XY triad plane. You
can also hold down the left mouse button on a plane and drag the point.

Figure 11-21.
The text boxes in
the **3D Move/Rotate**
dialog box change
based on the
selected triad item.

Unrestricted Move

Move in XY Plane

Move Along X Axis

Rotate Around Z Axis
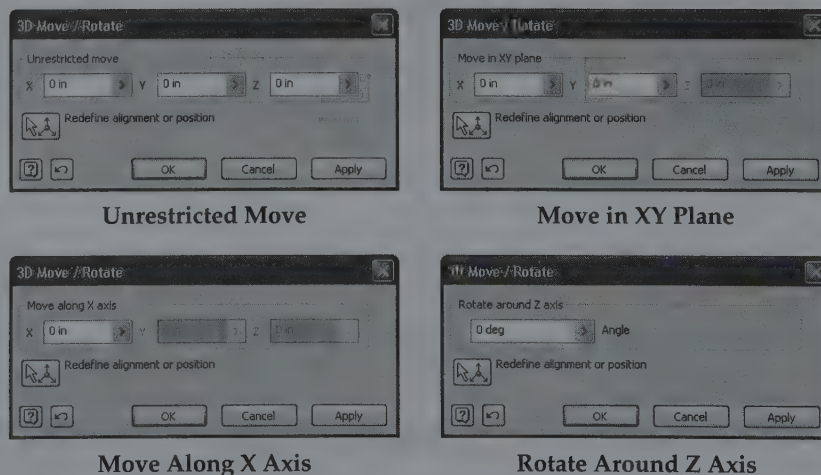
Axis Move

The **Axis move** option, available when you pick a triad arrowhead, allows you to move the grounded work point anywhere along the selected axis. Pick an arrowhead and specify values using the corresponding text box in the **3D Move/Rotate** dialog box. For example, the **Z** text box is available if you pick the Z triad axis. You can also hold down the left mouse button on an arrowhead and drag the point.

Axis Rotation

The **Axis rotation** option, available when you pick a triad axis, allows you to rotate the triad around the selected axis. The main purpose of an axis rotation is to adjust the position of the triad in order to move the point along a plane or axis in the new direction. Pick an axis and adjust the rotation using the **Angle** text box in the **3D Move/Rotate** dialog box. You can also hold down the left mouse button on an axis and drag to rotate the triad.

Finalizing Placement

Right-click and select **Tentative Drag** to display the previous position of the triad for reference. To return to the previous location and undo a movement, pick the **Undo** button. You may have the option of selecting the **Redefine alignment or position** button or right-clicking and selecting **Redefine** to reposition the work point at a specified location. If you select the triad sphere, you can relocate the grounded work point by picking a point. If you pick a triad axis, you can align the axis with a selected edge, axis, or sketched line. If you choose a triad plane, you can relocate the plane by picking a face or plane. Pick the **Apply** button or right-click and select **Apply** to create the grounded work point and remain in the **Grounded Work Point** tool. Pick the **OK** button or right-click and select **OK** to create a grounded work point and exit the tool.

> **NOTE**
>
> Another option for creating a grounded work point is to place an ungrounded work point using the **Work Point** tool. Then right-click on the ungrounded work point and pick **Ground**. To reposition the converted work point, right-click on the grounded work point and pick **3D Move/Rotate**.

Exercise 11-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-10.

The default *coordinate system* is fixed and universal for all objects in 3D space. You can view the default coordinate system *center point*, planes, and axes by expanding the **Origin** folder of the browser. You have, and will continue to, use the default origin content to develop models. If the default origin features, or existing model points, faces, planes, edges, or axes are insufficient for model development, one option is to use work features. An alternative is to specify a *user coordinate system (UCS)*. As you learn to use a UCS, it may help to consider a UCS as a coordinate system with the same items found in the **Origin** folder of the browser, but at a location and rotation of your choice.

Access the **User Coordinate System** tool to establish a UCS. The UCS triad and *heads-up display (HUD)* appears in the graphics window. See **Figure 11-22**. Depending on the current situation, the HUD displays a prompt, text boxes that you can navigate by pressing [Tab], or other information.

One option to locate the UCS is to enter X, Y, and Z coordinate values in the HUD, or pick anywhere in 3D space. This is an example of an unrestricted move operation. You can then use the triad and HUD to adjust the UCS. Apply similar unrestricted planar, axis move, and axis rotation techniques to adjust the UCS as you would to

coordinate system: The system of XYZ coordinate values that defines the location of points in 3D space.

center point: The intersection point of the X, Y, and Z axes in 3D space, or 0,0,0.

user coordinate system (UCS): A custom XYZ coordinate system with a center point and corresponding planes and axes at 90° to each other.

Ribbon
Model
> Work Features

UCS
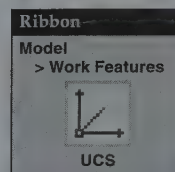
heads-up display (HUD): A temporary control bar that allows you to adjust UCS settings by entering values at the keyboard.

Figure 11-22.
The user coordinate system (UCS) triad and heads-up display (HUD) appear when you access the **User Coordinate System** tool.

UCS positioning triad

Heads-up display (HUD)

Specify origin of UCS | Absolute | ΔX | -0.301 in | ΔY | 2.705 in | ΔZ | -0.377 in

Unrestricted Move (Initial Placement)

Pick or drag to apply an axis move

Specify origin of UCS | Absolute | ΔY | 2.833 in

Axis Move

Pick or drag to apply an axis rotation

Specify rotation angle about Y axis | Relative | Δ< | 0.00 deg

Axis Rotation
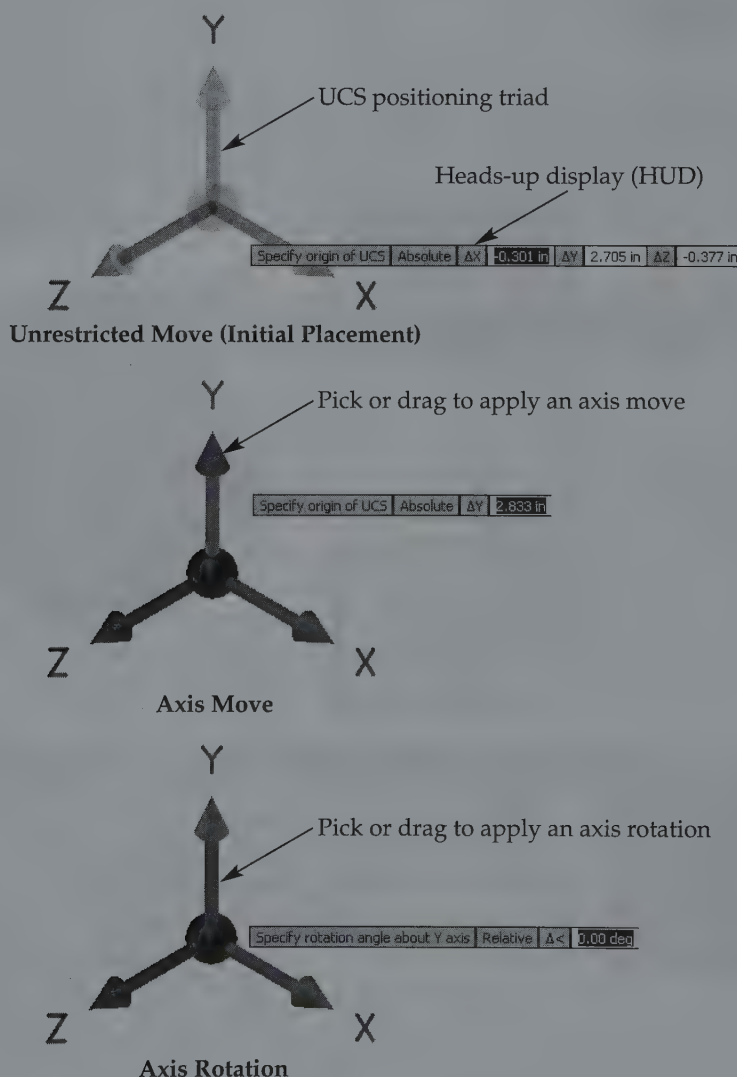
position a grounded work point. The main differences are that you enter values in the HUD instead of a dialog box, do not have to drag an unrestricted move, and cannot apply a planar move. To undo all movement and restart the tool, right-click and select **Restart**. Press [Esc] or right-click and select **Done** to cancel. Press [Enter] or right-click and choose **Finish** to accept the current orientation.

Another option for positioning a UCS is to pick a work point or a point on an existing feature or sketch to specify the UCS origin, followed by a point that defines the direction of the UCS X axis, and then a point to set the direction of the UCS Y axis. See **Figure 11-23**. Once you create a UCS, the UCS appears in the browser, as shown in **Figure 11-23**. The UCS axes are visible by default, and the UCS triad appears in the model. Use the UCS center point, planes, and axes as you would the content in the **Origin** folder of the browser.

NOTE

You can reference a UCS center point while sketching without projecting the center point onto the sketch plane.

Exercise 11-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 11-11.

Figure 11-23.
An example of picking points on an existing feature to locate a UCS. Notice the UCS in the browser and the difference between the default coordinate system and the UCS.
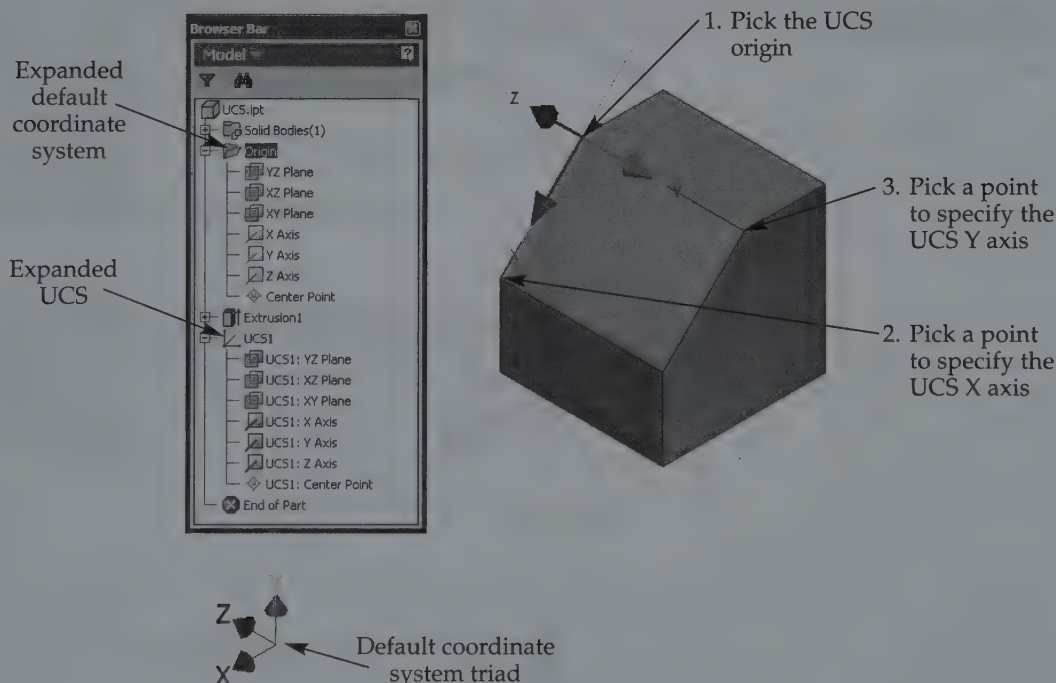
Figure 11-24.
Options that might be available when you right-click on a work feature in the graphics
window or browser.

Option	Description
Visibility	Displays the selected work feature or UCS triad. Deselect this option to hide items when necessary for clarity.
Consume Inputs	Causes the selected work feature to consume reference work features (those used to position the parent feature). Expand the parent to display and select consumed features.
Auto-Resize	Resizes the plane or axis past the extents of features hiding the plane or axis.
Flip Normal	Flips a displayed side of a plane. Depending on display settings, the "front," or positive, side of a plane is a lighter color than the "rear," or negative, side. For some modeling requirements, such as creating decal features, it may be critical to work on the appropriate side of a plane to prevent items from appearing reversed. You can also use **Flip Normal** when the positive or negative work plane offset is incorrect.
Adaptive	Allows the selection to adapt to changes made to associated assembly content.
Ground	Grounds the selection; used primarily to convert work points to grounded work points.
Show Inputs	Highlights geometry such as points and edges selected to form the work feature.
Export Object	Marks the selection for export when you export a derived component.

Work Feature Adjustments

All work features, including those in the **Origin** folder of the browser and those associated with a UCS, include basic adjustment options. You can adjust the *displayed* position of a work plane by hovering the cursor over an edge until you see the **Move** icon. Then hold down the left mouse button to drag the work plane to the desired location. To change the *displayed* size of the work plane or work axis, hover the cursor over a work plane corner or work axis end until you see the **Resize** icon. Then, hold down the left mouse button to drag the corner or end to the desired size. **Figure 11-24** describes options available when you right-click on a work feature or UCS triad in the graphics window or browser, depending on the selected item.

NOTE

To relocate a work feature or UCS in space, you must right-click on the work feature or UCS in the graphics window or browser and select **Redefine Feature**. You can then recreate the item. When you redefine the location of a grounded work point, the grounded work point reverts to an ungrounded work point.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. Briefly describe work features and explain their use.
2. What is a work plane?
3. Explain the concept of infinite work planes.
4. What tool allows you to project the intersection of nonplanar feature faces and the work plane for sketch development?
5. Briefly explain when you should use the **Project Cut Edges** tool instead of the **Project Geometry** tool to project geometry.
6. Describe a work axis.
7. Identify at least three uses for work axes.
8. What is a work point?
9. Identify at least three uses for work points.
10. Explain the difference between a grounded and an ungrounded work point.
11. What is the purpose of the triad that appears with the **3D Move/Rotate** dialog box when you create a grounded work point?
12. Define the term *UCS* and explain when you might establish a UCS.
13. Describe how to adjust the displayed position and size of work plane.
14. Explain why it might be necessary to flip a work plane.
15. How can you physically relocate a work feature or UCS in space?

Problems

1–3 Instructions:

- *Open the specified part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **File:** P9-1.ipt
   Save as: P11-1.ipt
   Title: SCREW
   Specific Instructions: Create a work plane tangent to the screw head by selecting the tangent face and the appropriate default plane, as shown. Open a sketch on the work plane and sketch the geometry shown. Place a 4.85 mm through hole at the sketched **Point, Center Point**. The final part should look like the part shown.

Projected axis

Point, Center Point

Projected edge

4.85 thru hole
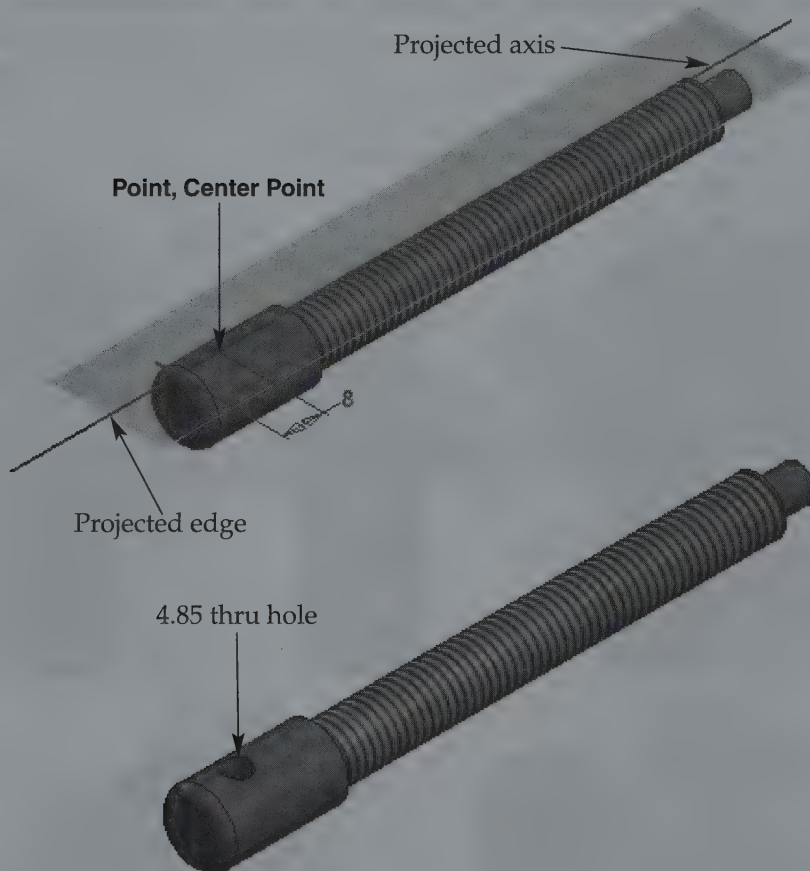
2. **File:** P10-3.ipt
   Save as: P11-2.ipt
   Title: HUB
   Specific Instructions:
   A. Offset a work plane 2.25″ from the XY plane. Open a sketch on the work plane and sketch the geometry shown in A. Convert the projected cylinder edge to the construction format.
   B. Extrude the sketch in the negative direction using the **To Next** termination option. Add the round shown in B using the **Full Round Fillet** option (the work plane visibility is off for clarity).
   C. Open a sketch on the YZ plane and sketch the geometry shown in C. Convert the lower projected line to the construction format. The diagonal line and the upper projected line should use the sketch geometry format.
   D. Use the sketch to create a .25″ thick rib.
   E. Use the **Circular Pattern** tool to pattern the extrusion as shown in E. Select the default Y axis as the rotation axis and specify a count of 3, a fitted position method, and an angle of 360°.

Projected cylinder edge (convert to construction format)

1.000
.500
1.000
.500
45.00

A   B   C

Full round

D   E
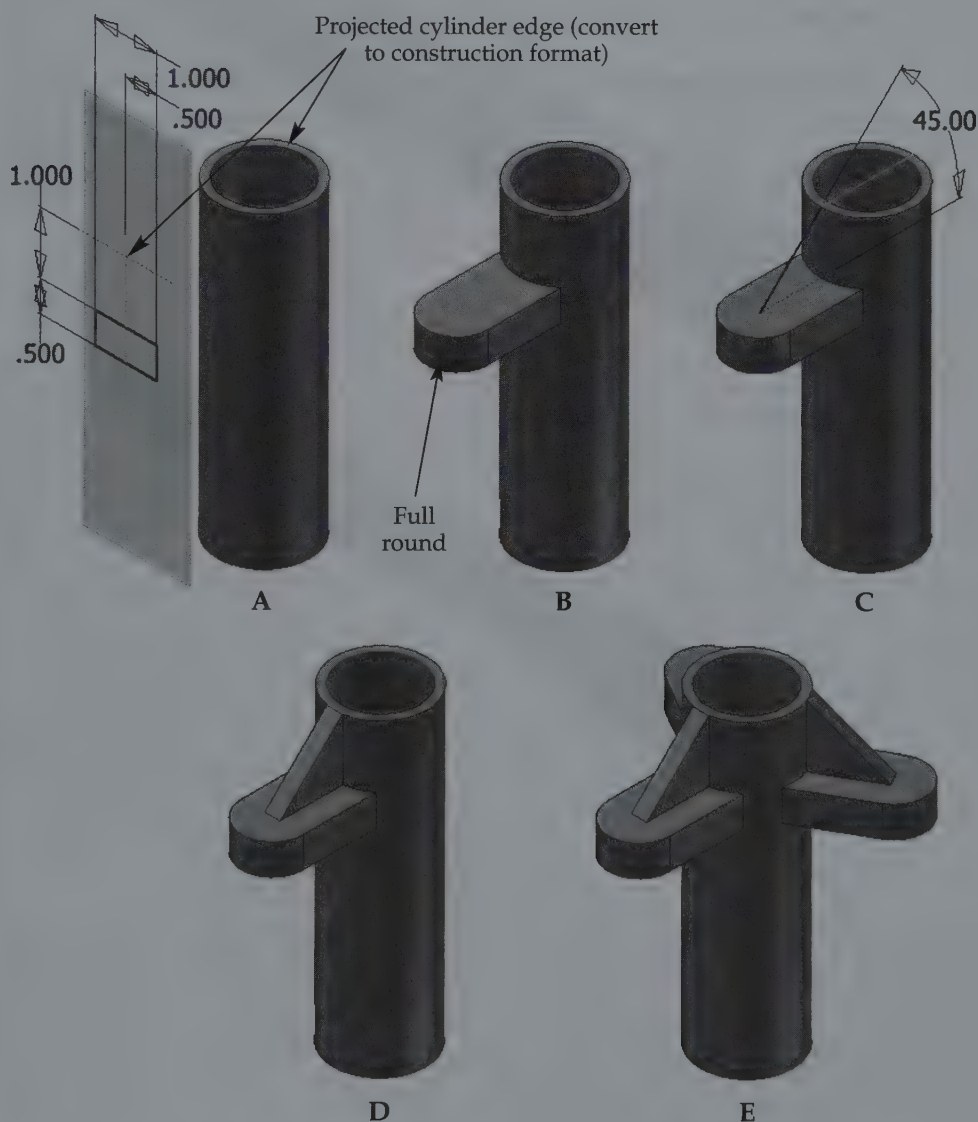
3. **File:** P10-9.ipt
   Save as: P11-3.ipt
   Title: Change the title to 45° SPLIT ELBOW
   Part Number: Change the part number to IAA-020-02
   Specific Instructions:
   A. Create a work plane by selecting the XY plane and Extrusion1, as shown in A. Open a sketch on the work plane, and sketch the geometry shown.
   B. Extrude the sketch toward the elbow using the **To Next** termination option, as shown in B.
   C. Create a Ø50 mm circle on the face of Extrusion3, concentric to Extrusion3, and extrude the profile 8 mm away from the elbow, as shown in B.
   D. Add a Ø25 mm hole beginning on the face of Extrusion4 and terminating at the inside face of the 45° elbow, as shown in C.
   E. Add a M8X1.25-6H full depth thru hole 19 mm vertically aligned with the center of Hole3, as shown in C. Create a work axis by selecting Hole3.
   F. Use the **Circular Pattern** tool to pattern the hole, as shown in D. Select Work Axis1 in the browser as the rotation axis and specify a count of 6, a fitted position method, and an angle of 360°. Add three 5 mm fillets as shown.

Projected Z axis

Projected cylinder edge

Extrusion1

A

Extrusion4   Extrusion3

B

Hole4

Hole3

C

Fillets

D

4–6 Instructions:

- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*

- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

4. **Title:** BASE
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-021-01
   Project: BASE
   Material: ABS Plastic
   Color: Gray (Dark)
   Save as: P11-4.ipt
   Specific Instructions: Create the sketch shown in A on the XY plane. Extrude the sketch 8″ midplane. Shell the extrusion .125″, removing the bottom face, as shown in B. Offset a work plane –.25″ from the bottom face, as shown in C. Open a sketch on the work plane and sketch the lines shown. Use the **Rib** tool to create the rib network shown in D.

Work plane

2.000

0.500

4.000

A

B

1.875

1.875

Ribs are .125″ thick

Rib network is offset .25″ from the bottom of the part

C

D
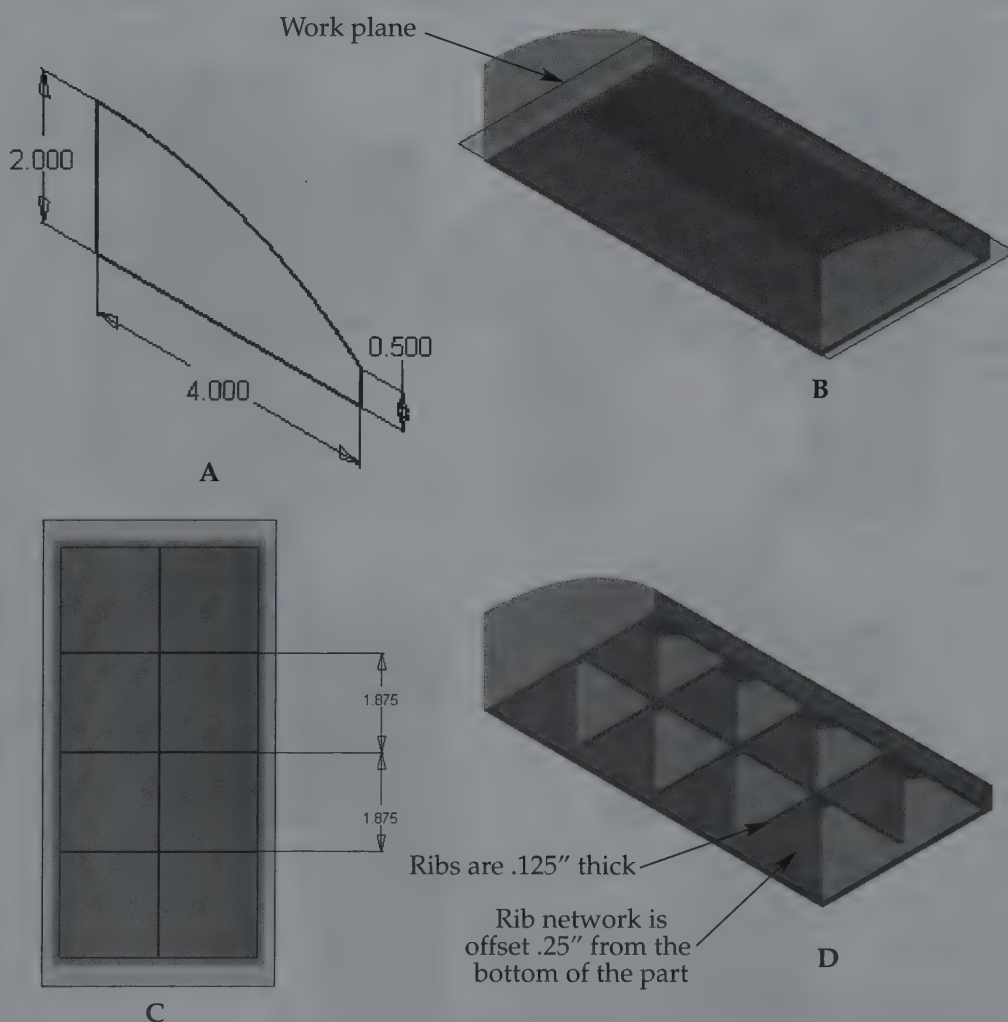
5. **Title:** SHAFT
   Units: Metric
   Template: Part-mm.ipt
   Part Number: IAA-022-01
   Project: AIRCRAFT
   Material: Default
   Color: Default
   Save as: P11-5.ipt
   Specific Instructions:
   A. Sketch a Ø20 mm circle on the XY plane. Extrude the sketch 60 mm midplane. Sketch the geometry shown in A on the left face of the cylinder. Share the sketch.
   B. Create Work Plane1 by picking the specified line and point. Sketch the geometry shown in B on the work plane.
   C. Cut-extrude the sketch 5 mm. Create Work Plane2 by picking the specified line and point. Sketch the geometry shown on the work plane.
   D. Extrude the sketch 5 mm. The final part should look like the part shown in D with all work features and Sketch2 hidden.

A

B

C

D
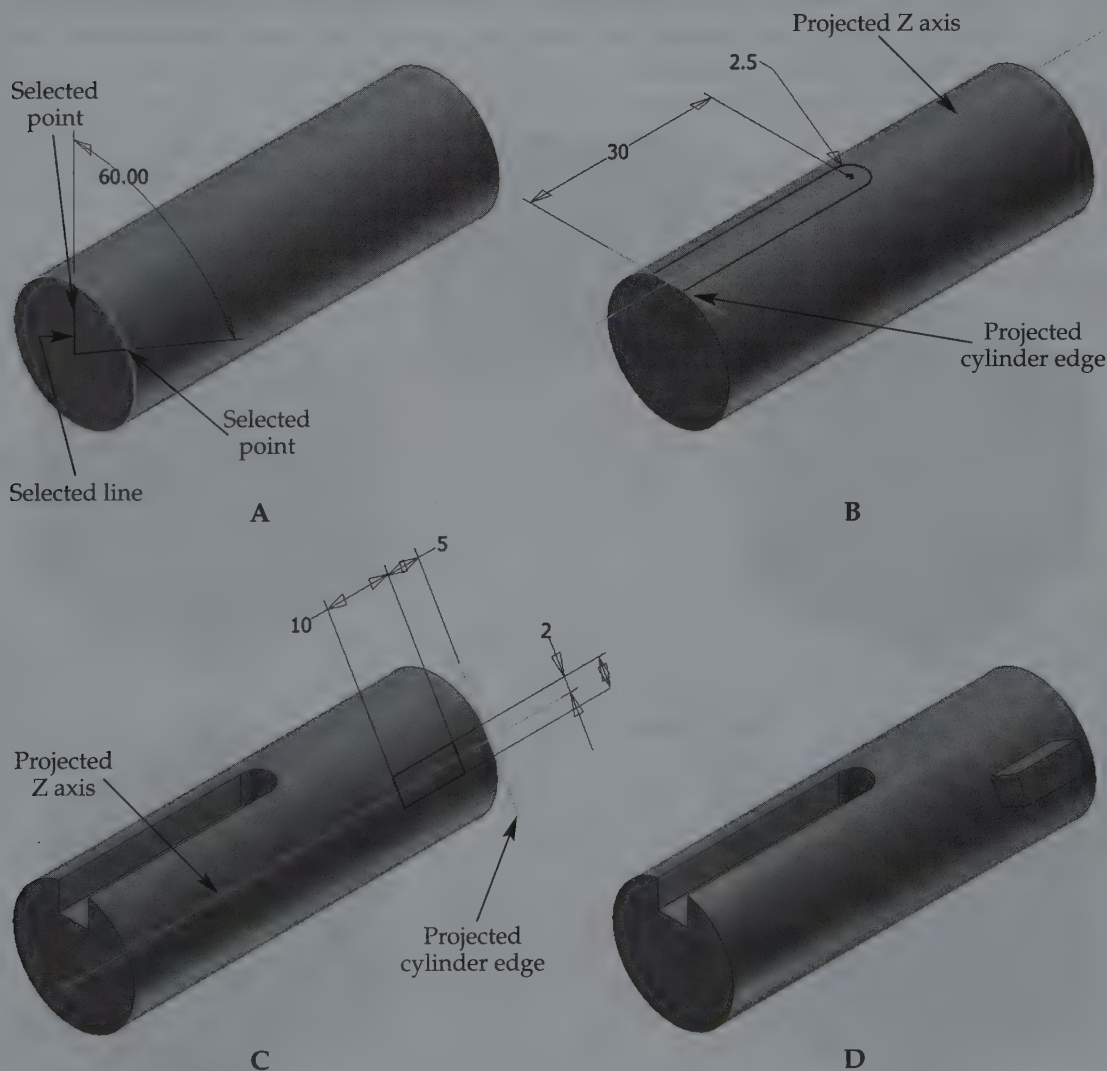
6. **Title:** BOX
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-023-01
   Project: BOX
   Material: ABS Plastic
   Color: As Material
   Save as: P11-6.ipt
   Specific Instructions:
   A. Sketch a 12″ × 12″ rectangle on the XY plane. Extrude the sketch 12″ in the positive direction. Shell the box .25″ to the inside, removing the top face. Sketch the geometry shown, on one of the box faces. Cut-extrude the sketch using the **All** extents option.
   B. Use the **Rectangular Pattern** tool to pattern Extrusion2, as shown in B. Select the horizontal edge shown to define Direction1, and specify a Direction1 count of 3 and a spacing of 4″. Select the vertical edge shown to define Direction2, and specify a Direction2 count of 4 and a spacing of 3″.
   C. Create Work Plane1 and Work Plane2 as shown in C and D by selecting exterior edges. Mirror Rectangular Pattern1 across Work Plane1. Mirror Rectangular Pattern1 across Work Plane2. The final part should look like the part shown in D with all work features hidden.

A

B

C

D
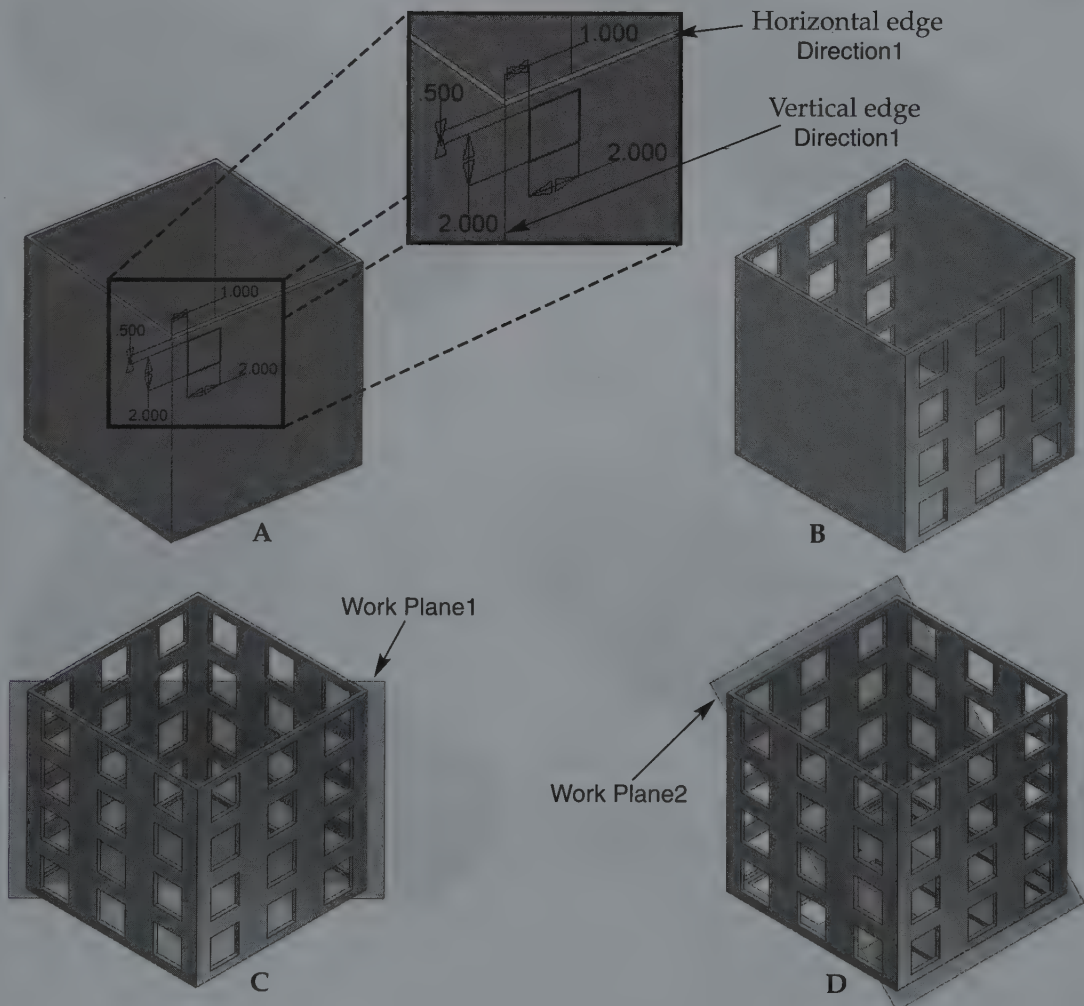
7. **Title:** MUG
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-101-01
   Project: MUG
   Material: Porcelain
   Color: As Material
   Save as: P11-7.ipt
   Specific Instructions:
   A. Go to the Student Web site (www.g-wlearning.com/CAD) and select **Chapter 11** in the **Chapter Materials** drop-down list. Download the Porcelain.styxml style definition file. Save the file in a location where you will be able to locate it later.
   B. Access the **Style and Standard Editor** and pick the **Import** button to display the **Import style definition** dialog box. Locate and select the downloaded Porcelain style definition file and pick the **Open** button. Assign the Porcelain material to the file.
   C. Open a sketch on the XZ plane and sketch a Ø2″ circle with the center coincident to the projected center point. Extrude the sketch 2.5″ in the positive direction.
   D. Offset a work plane –1.5″ from the XY plane. Offset another work plane –.75″ from the XY plane. Offset a third work plane –.75″ from the top face, as shown in A.
   E. Create Work Axis1 shown in A by selecting Work Plane1 and Work Plane3. Create Work Axis2 shown in A by selecting Work Plane2 and Work Plane3.
   F. Open a sketch on Work Plane3 and sketch the rectangle shown in B.
   G. Revolve the sketch around Work Axis1, 90° in the positive direction, as shown in C. Share the revolution sketch and revolve the profile around Work Axis2 counterclockwise using the **To Next** extents option, as shown in C.
   H. Open a sketch on the revolution face shown in C and project edges. Extrude the sketch using the **To Next** extents option, as shown in D.
   I. Hide all work features. Add a .25″ shell removing the top cylinder face. Add a .125″ equal-distance chamfer to the outside edge of the bottom cylinder. Add .05″ and .1″ fillets and rounds as shown.

Work Plane1: 1.5″
away from YZ plane
(−1.5″)

Work Plane2: .75″
away from YZ plane
(−.75″)

Projected
Work Axis1

Work
Axis1

Graphics
sliced for
clarity

Work
Axis2

.500

.250

.500

Work
Plane3: .75″
below top
face (−.75″)

Projected
center
point

Midpoint
horizontal

Z   X

Y

Z   X

Y

A

B

Sketch of
projected edges

First
revolution

.1″ fillets
and rounds

.05″ rounds

Second
revolution

.1″ fillet
(inside edge)

.05″ rounds

.125″ × .125″
chamfer

C

D

Lofts, Sweeps, and 3D Sketching

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create lofts.
✓ Create sweeps.
✓ Develop 3D sketches.

This chapter describes tools and applications for creating lofts and sweeps. Lofts and sweeps are sketched features that are often more complex than those you create using other common single-sketch or placed feature tools. This chapter also explains 3D sketching, which expands your ability to create features.

Lofts

A *loft* forms a feature from two or more *sections*. **Figure 12-1** shows an example of a beverage glass built using a loft between square and circular sections. Loft sections independent of existing geometry are most often sketches. You can also use existing feature faces as loft sections without creating a sketch of referenced edges. The browser identifies face sections as Edges. For some applications, a combination of different types of sections is appropriate. See **Figure 12-2**.

You usually create sketched loft sections on work planes or existing planar feature faces. For example, the circular section in **Figure 12-1** is a sketch on a work plane offset from the default XZ plane. Another example is the circle in **Figure 12-2**, which was sketched on the default YZ plane between the existing extrusions. The multiple selections required to build a loft require planning, and sketch placement is especially important.

loft: A feature that references and blends two or more sections located on different planes.

sections: Sketches and existing feature faces used to develop loft features.

> **NOTE**
>
> There is no limit to the number of loft sections you can use. Loft section sketches must be closed loops unless you want to create a surface loft. Loft sections can be at any angle and can be located anywhere in space from other sections; they do not have to be parallel.

Figure 12-1.
A beverage glass created by lofting a square sketched on the default XZ plane to a circle sketched on a work plane offset above the XZ plane. A shell, fillets, and rounds complete the part.

Circular sketched section

3.000

Square sketched section

Loft blends to form a feature
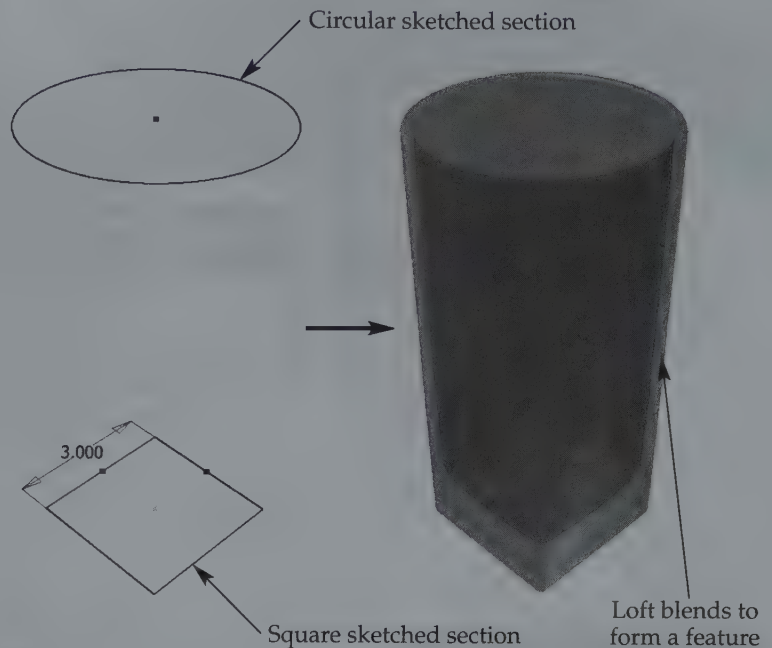
Figure 12-2.
A—Extruded ellipse faces selectable as loft sections. B—A combination of feature faces and a sketch profile used to create a loft. C—A cabinet handle further developed using a loft from a feature face to a sketched circle to another feature face.

Sketched section

.625

1.500

Feature face sections

A　　　　　B　　　　　C

Ribbon
Model
> Create

Loft

Type
[Ctrl]+[Shift]+[L]
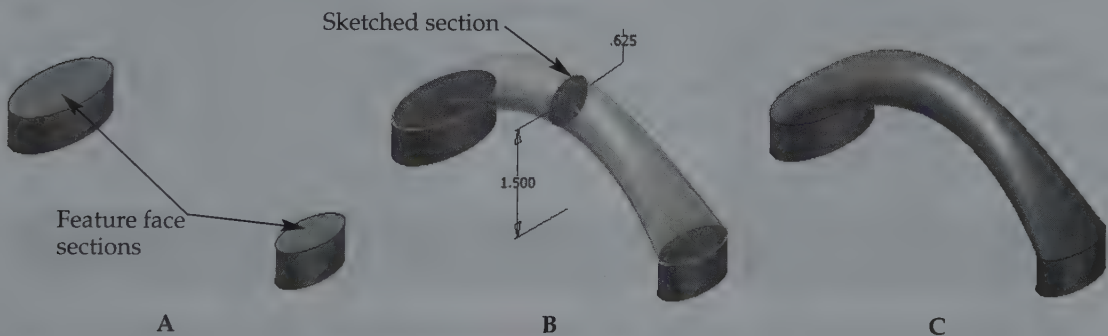
Access the **Loft** tool to create a loft using the **Loft** dialog box. See **Figure 12-3**. Fewer options are available for creating a base feature than for building additional features. Begin defining loft parameters by picking sections. You may be able to select each section using repeated picks. If not, or if you exit section select mode, pick the **Click to add** button in the **Sections** list box and continue to select sections. Pick sections in a logical, sequential order to ensure that the loft forms correctly. This is especially important when lofting between two or more sections and when it is necessary to adjust conditions and transition, as explained later in this chapter. Selected sections are listed in the **Sections** column, and a preview of the loft appears by default.

NOTE

If more than one possible sketch profile is available on a section sketch plane, you may have to pick the sketch and then pick the desired sketch profile.

Figure 12-3.
The **Curves** tab of the **Loft** dialog box.

Rails radio button

Center Line radio button

Area Loft radio button

Enable/disable feature preview
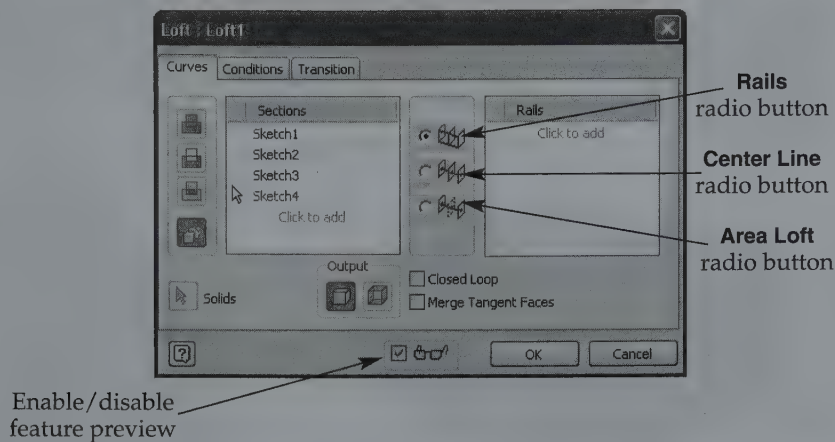
PROFESSIONAL TIP

For clarity, it may help to pick loft sketches from the browser.

You have the option of lofting closed profile sections as a solid or surface by picking the appropriate button in the **Output** area. You can loft open profile sections only as a volumeless surface. When you create a solid base feature, the only possible operation is to form a new solid body, as evident by the selected **New Solid** button. When you add a loft, the **Join**, **Cut**, and **Intersect** operation buttons are available for adding, removing, or combining material using the loft.

To create a basic loft, you may now be ready to pick the **OK** button. For lofts that are more complex, the **Loft** dialog box includes several shape control and appearance options. Select the **Closed Loop** check box to place a *closed loop control*. In some cases, abrupt edges form between loft sections, resulting in an unacceptable shape. If your design requires a smooth loft, you may be able to remove the abrupt edges and create the desired shape by selecting the **Merge Tangent Faces** check box.

closed loop control: A loft operation that fixes the first and last section together; available only when using rails.

Exercise 12-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 12-1.

Rails

A *rail* is a guide that lofts follow during blending. See **Figure 12-4**. A rail can be a 2D or 3D sketched curve and must intersect a point on each loft section. You can create as many rails as needed to establish the desired loft shape. The following steps represent a typical approach to creating a loft using 2D rails:

rail: A 2D or 3D sketched curve used with sections to manipulate and further control the loft shape.

1. Add a work plane if an appropriate sketch plane on which to sketch the rail is not available.
2. If necessary, add work points on each section to help define the location where the rail intersects each section. Work points are not necessary if you plan to select existing sketch curve points.
3. Open a sketch on the plane described in Step 1.

Figure 12-4.
Examples of models lofted from the same sections to illustrate the impact of adding a rail. A typical loft without a rail blends sections together from one section to another, along the straightest path between the sections.

Loft without a Rail   Loft with a Rail
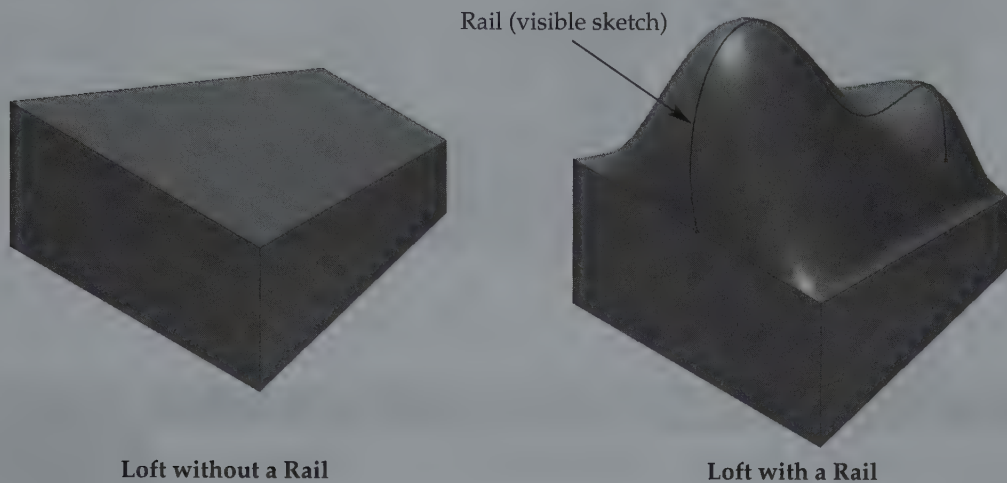
4. Project sketched section points, such as curve endpoints or previously created work points, onto the sketch plane to define the location at which the rail intersects each section.
5. Use sketch curve tools to create the rail. Be sure the rail intersects the projected point on each of the sketched sections. See **Figure 12-5**.
6. Repeat Steps 1 through 5 as needed to create additional rails.
7. Access the **Loft** tool, pick the sections, and select the **Rails** radio button. Pick rails or use the **Click to add** button in the **Rails** list box to select rails if necessary. Selected rails list in the **Rails** column, and a preview of the loft appears by default.

Figure 12-5.
Creating a 2D loft rail.
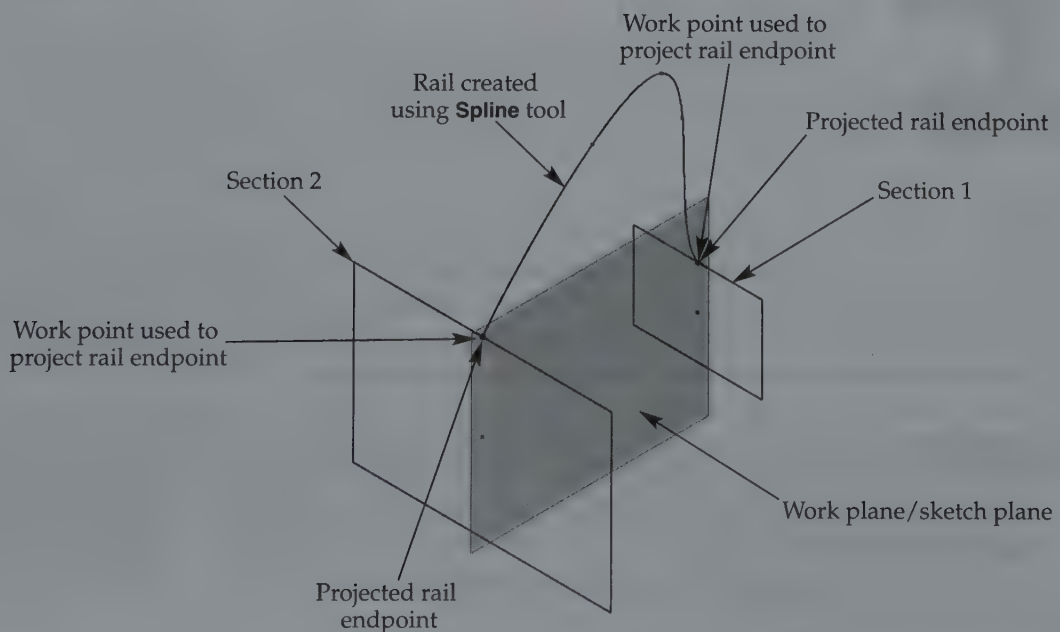
Rails should be as smooth and uncomplicated as possible. You may not be able to generate a loft that contains very abstract rails.

Centerline

A *loft centerline* is similar to a rail, but occurs along the center path of the loft operation. See **Figure 12-6**. Typically, a centerline is constrained to the center of sections. However, unlike rails, a centerline does not need to intersect sections. Once you add a centerline, access the **Loft** tool, pick the sections, and select the **Center Line** radio button. Pick the centerline or use the **Select a Sketch** button in the **Center Line** list box to select the centerline if necessary. The selected centerline is listed in the **Center Line** column, and a preview of the loft appears by default.

loft centerline: A rail that acts as a path for blending sections along and symmetrically around the centerline sketch.

Figure 12-6.
Using a 2D loft centerline to control the shape of a loft.

Loft sections

Centerline

The default straight-line transition between sections without using a centerline

Loft transition controlled by a centerline to create the desired feature
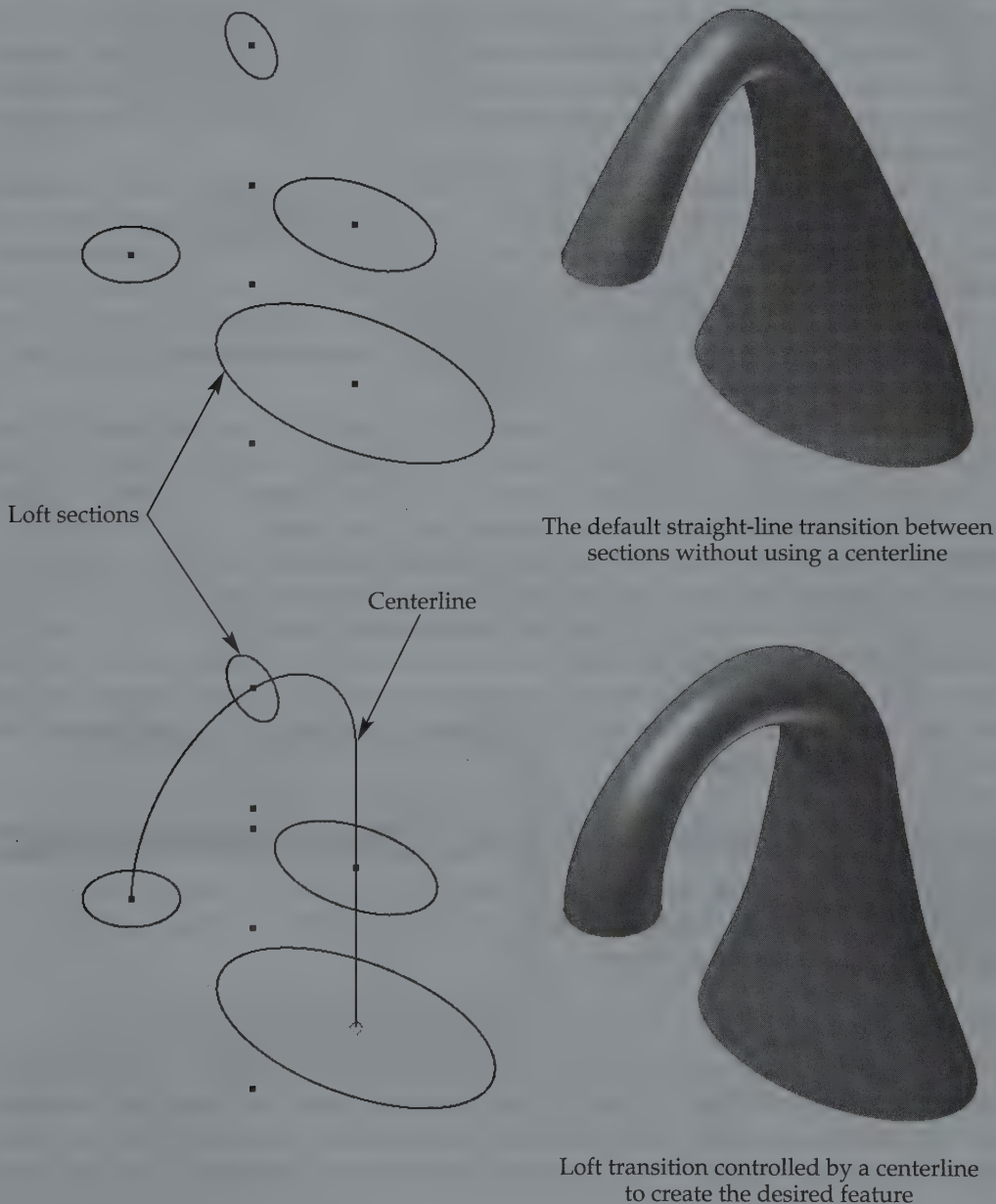
Exercise 12-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 12-2.

Area Lofts

placed sections: Loft sections created without a sketch and placed along a selected centerline; calculated based on the loft cross section at the selected location.

An area loft uses a centerline and allows you to modify the cross-sectional area of selected sections and additional *placed sections* to adjust the size and shape of the loft. Create loft sections and a centerline exactly as you would to loft with a centerline. Then access the **Loft** tool, pick sections, and select the **Area Loft** radio button.

Pick the centerline or use the **Select a Sketch** button in the **Center Line** list box to select the centerline if necessary. A preview of the loft appears, along with the position and cross-sectional area of each selected section. See **Figure 12-7A**. You can now pick points along the centerline to place sections if necessary. See **Figure 12-7B**. If you exit the placed sections select mode, select the **Click to add** button in the **Placed Sections** list box to select the placed sections.

The **Section Dimensions** dialog box, shown in **Figure 12-7B**, appears when you define a placed section. The **Driving Section** radio button is active by default and allows you to change the cross-sectional area using the options in the **Section Size** area. Pick the **Driven Section** radio button to display the section for reference only and disable the **Section Size** area.

The options in the **Section Position** area allow you to locate placed sections precisely. The beginning of the loft, at the first selected section, is at a position of 0. The end of the loft, at the last selected section, is at a position of 1. The **Proportional Distance** radio button is active by default and allows you to specify a location in the text box based on a fraction between 0 and 1. For example, enter .5 to place the section halfway between the start and end of the loft. Pick the **Absolute Distance** radio button to specify a distance from 0 in the text box.

The **Section Size** area controls the cross-sectional area of the section, which influences the shape of the loft. Increasing a section area "bulges" the loft, while reducing the section area indents the loft. See **Figure 12-7C**. The **Area** radio button is active by default, allowing you to specify a cross-sectional area value in the text box. Pick the **Scale** radio button to specify a scale factor in the text box. The default scale is 1. Increase or decrease the scale to increase or decrease the section area.

Pick the **OK** button to exit the **Section Dimensions** dialog box. You can modify the dimensions of any section, including non-placed sections, by double-clicking on the dimensions or right-clicking on the dimensions and selecting **Edit**. Delete a placed section from the **Placed Sections** list box by pressing [Delete], or right-click on the dimensions and pick **Delete**.

NOTE

You cannot modify the position of non-placed sections using the **Section Dimensions** dialog box. The area of non-placed sections is initially set to **Driven**.

Conditions

The **Conditions** tab, shown in **Figure 12-8**, allows you to adjust the boundary characteristics of start and end sections, some rails, and point sections. Select an item from the list and make changes to the default condition by choosing an option from the **Conditions** flyout. **Figure 12-8** shows conditions that might be available from the flyout.

Figure 12-7.
A—Create an area loft using the same methods as a centerline loft. B—Placing and adjusting loft sections to modify the loft shape. C—A completed area loft with one placed section. Notice the difference in shape at the placed section from the preview shown in A and B.

Position = (Start)
Area = (0.1360849 in^2)

Centerline

.500

Position = (End)
Area = (0.1739092 in^2)

Selected sections
with corresponding
area dimensions

1.375

.500

A

Proportional
Distance

Driving
Section

Driven
Section

Area

Section Dimensions

Section Position

0.619288

Section Size

0.1583708 in^2

OK   Cancel

Absolute
Distance

Scale
Factor

Placed section with
default area dimensions

Placed section
with smaller-than-default
cross-sectional area

C

Position = (Start)
Area = (0.1360849 in^2) Position = 0.619288
Area = 0.1583708 in^2

.500

Position = (End)
Area = (0.1739092 in^2)

1.375

.500

B

Figure 12-8.
The icons shown in the **Conditions** tab of the **Loft** dialog box adjust boundary conditions.

Icons that may be found in the
Conditions drop-down list

Loft : Loft1

Curves   Conditions   Transition

		Angle	Weight
Sketch1 (Section)		90.00 deg	0.000 ul
Sketch2 (Section)		90.00 deg	5.000 ul

OK   Cancel

Free Condition

Tangent (G2) Condition

Direction Condition

Smooth (G2) Condition

Tangent

Tangent to Plane

Sharp Point
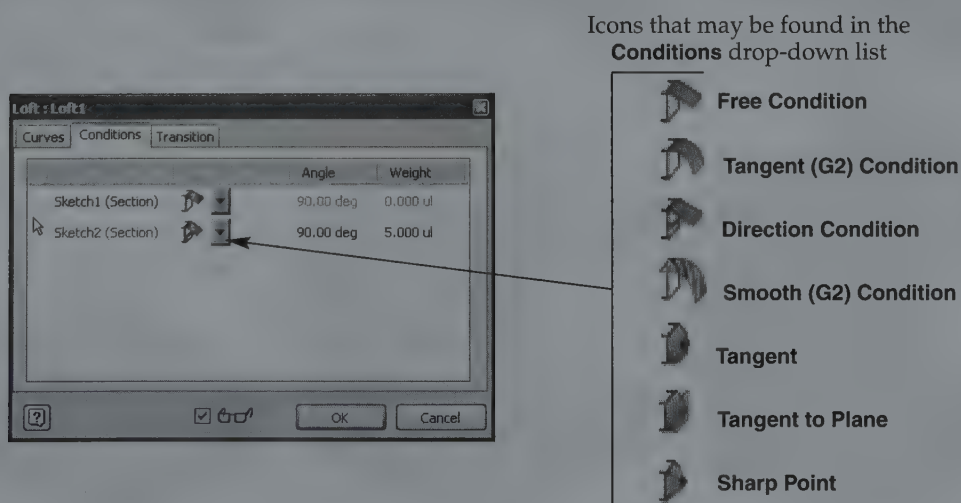
The **Free Condition** option is default and applies a 90° angle and a weight of 0. Pick the **Direction Condition** button to apply a direction condition to a 2D sketch profile section. Use the **Weight** text box to specify a unitless loft weight, from 0 to infinite, to further define the appearance of the loft. Weight is relative to the size of the loft feature. A weight too high may result in an undesirable loft shape. When you enter a small weight value, a more direct transition occurs between sketch profiles. See **Figure 12-9**. When you enter a larger number, a more indirect, meandering transition occurs between sketch profiles. The **Angle** text box is available when you enter a weight greater than 0 and allows you to specify the angle between the faces that the loft creates and the selected sketch profiles. See **Figure 12-10**.

The **Tangent (G1) Condition**, and **Smooth (G2) Condition** options are available when a section is a feature face, a sketch on a feature with projected face edges, or a sketch adjoining a lateral surface or body. Select the **Tangent (G1) Condition** option to place a tangent constraint between the section and the planar face on which you created the section. Select the **Smooth (G2) Condition** option to place a smooth constraint between the section and the planar face on which you created the section. Specify a unitless loft weight in the **Weight** text box to control the appearance of the loft, much like the tangent condition.

The **Sharp Point**, **Tangent**, and **Tangent to Plane** options are available when a section is a point. The default **Sharp Point** condition creates a direct loft from a section to a point, without boundary condition. See **Figure 12-11A**. Select the **Tangent** option to create a rounded point by applying a tangent constraint between the planar section and point. Use the **Weight** text box to specify a unitless loft weight. A higher weight value creates a rounder point section. See **Figure 12-11B**. Select the **Tangent to Plane** option to create a rounded, but flattened point based on the orientation of a selected

Figure 12-9.
These models were identical until the loft weight was changed.

Weight = 0   Weight = 10   Weight = 25

Figure 12-10.
Examples of the effect of changing the angle on loft formation.

90° angle   30° angle   30° angle

90° angle   90° angle   30° angle

Free Condition   **Direction Condition**   **Direction Condition**
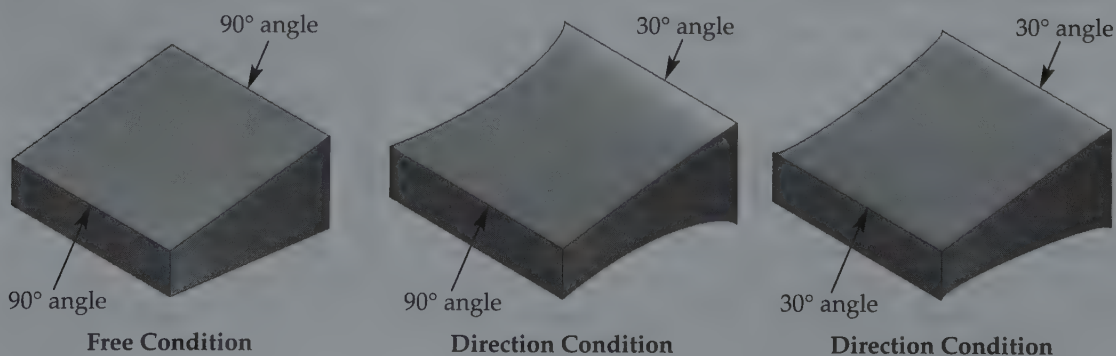
Figure 12-11.
A—A loft created using a point as a section and the **Sharp Point** condition. B—A **Tangent** condition with a weight of 4. C—A **Tangent to Plane** condition with a weight of 4.

Point section

Point section

Point section

Selected plane (face)

A

B

C
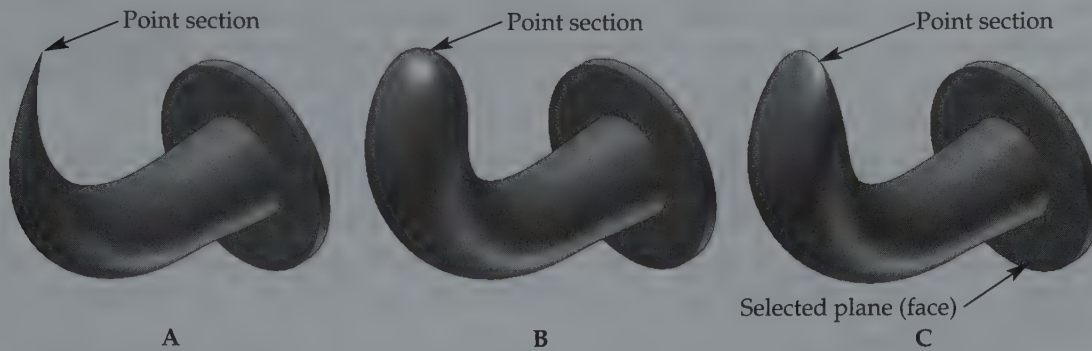
plane or face. After you pick the **Tangent to Plane** button, select a plane or face. Use the **Weight** text box to specify a unitless loft weight. A higher weight value creates a rounder and flatter point section. See **Figure 12-11C**.

Exercise 12-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 12-3.

Transition

The options on the **Transition** tab help ensure that complex lofts generate correctly by mapping transition points. See **Figure 12-12**. Inventor attempts to apply a logical transition by default with the **Automatic Mapping** check box. However, the automatic mapping function may select the incorrect transition points, causing the loft to build an incorrect feature. Manual point mapping usually takes care of this problem by allowing you to align sketch sections and correctly specify the loft path according to selected points on each section.

Figure 12-12.
An example of when automatic mapping, the default loft transition, creates an inappropriate feature. Notice the difference in shape with correct transition applied.

Loft Sections

Incorrect Default Loft Transition

Correct Modified Loft Transition
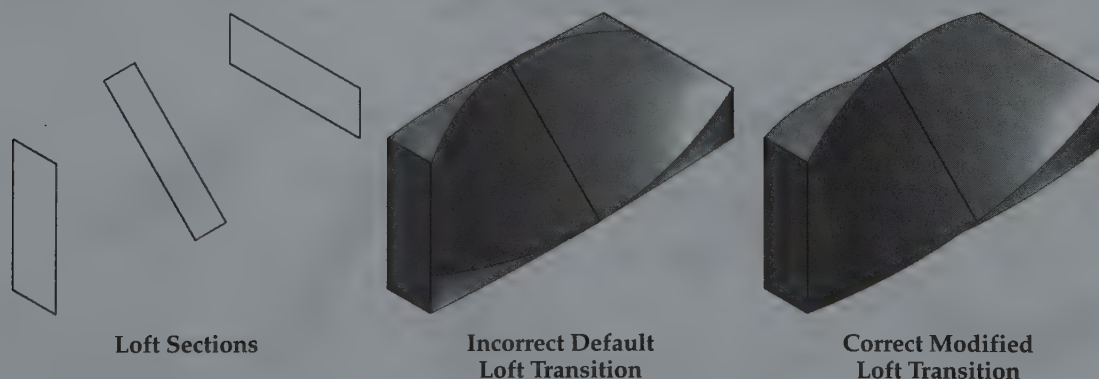
To map points on each section, deselect the **Automatic Mapping** check box to display all automatic transition paths, or point sets, in the **Point Set** column and in the graphics window. See **Figure 12-13A**. The **Map Point** column indicates the points associated with the selected point set, in the sequence of section selection. The **Position** column identifies the map point location relative to the line on which it occurs. A position of 0 defines one end of the line, and a position of 1 defines the opposite end of the line. For example, to place a map point in the middle of the line, enter a position of .5.

One option for redefining automatic map points is to pick the point set to modify from the **Point Set** column, which highlights the point set in the graphics window. You can then move the cursor over the map point to move, and without clicking on the point, move the point to the desired location and pick. Another option is to adjust specific map points by entering a new position for the points in the corresponding **Position** text box.

In some cases, if the loft is extremely twisted and numerous incorrect points have been calculated, it may be easier to remove all automatic point sets and create new sets, or to remove and modify all but one or two point sets. Pick a point set to remove and press the [Delete] key. To create a point set, pick the **Click to add** area in the **Point Set** list box. Then select a point on each section in sequential order. Usually, you can pick any available point on a sketch, but it is very important that you choose corresponding points on each sketch. See **Figure 12-13B**. Pick the **OK** button to generate the loft feature.

Figure 12-13.
A—Deselect the **Automatic Mapping** check box to see the list of point sets and to change the position of any incorrect point sets. B—Replacing existing point sets with new point sets. Use as many point sets as required to indicate the correct transition. Notice the incorrect map point positions shown in A compared to the correct map point positions shown in B.
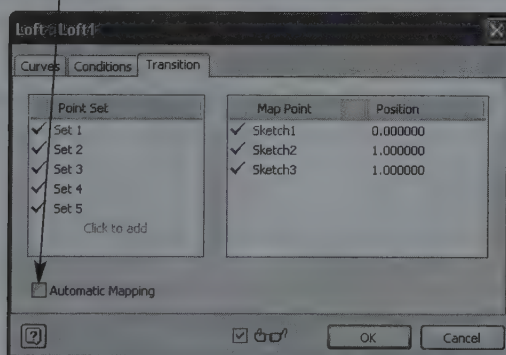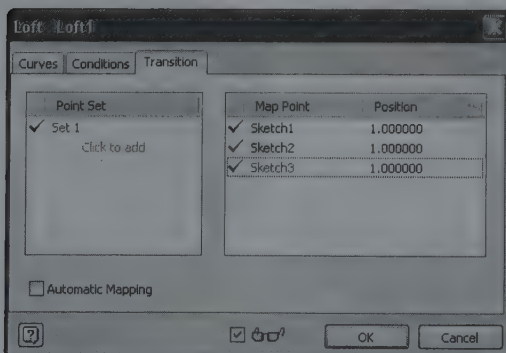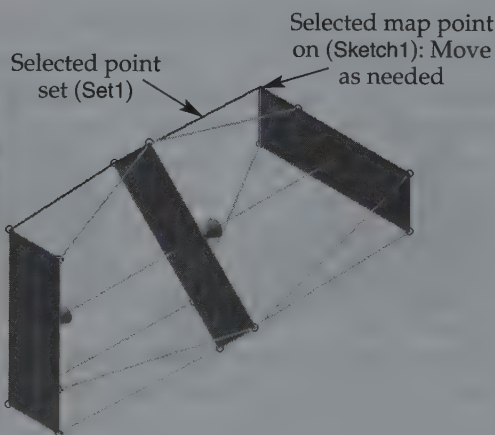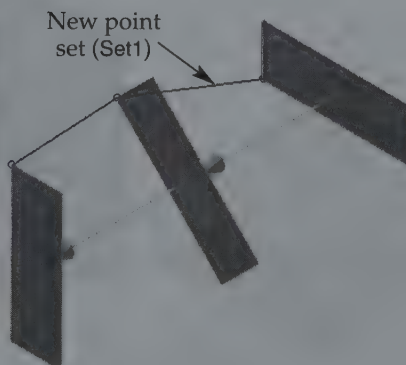
Exercise 12-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 12-4.

Sweeps

A *sweep* forms a feature by shaping a profile along a path. **Figure 12-14** shows an example of a tubular frame rail built using a sweep. A sweep requires a sketched profile, similar to the profile needed for an extrusion, and a sketched path. To ensure that a sweep generates without errors, the profile and path should intersect. Sometimes, an intersecting profile and path is not necessary, but Inventor will display a message asking if you want to proceed with the sweep operation. If the sweep does not form, modify the sketches so that the profile and path intersect.

sweep: A feature created by guiding, or sweeping, a sketch profile along a sketch path.

> **NOTE**
>
> Typically, a coincident constraint establishes a proper profile and path intersection. The profile and path do not need to be perpendicular, although a perpendicular relationship is common and often appropriate.

Access the **Sweep** tool to create a sweep using the **Sweep** dialog box. See **Figure 12-15**. The profile and/or path may be selected automatically when you create a basic sweep feature. If not, use the **Profile** and **Path** buttons and select the profile and path. The **Optimize for Single Selection** check box is active by default, aiding in the selection process. A preview of the sweep appears by default.

Ribbon
Model
> Create

Sweep

Type
[Ctrl]+[Shift]+[S]
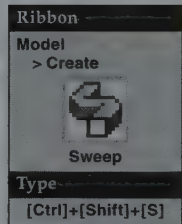
> **NOTE**
>
> If more than one possible sketch profile or path is available, you may have to pick the sketch and then pick the desired sketch profile or path.

Figure 12-14.
A tubular frame rail created by sweeping a circle sketched perpendicular to a sketched path of lines and arcs. Additional features complete the part.
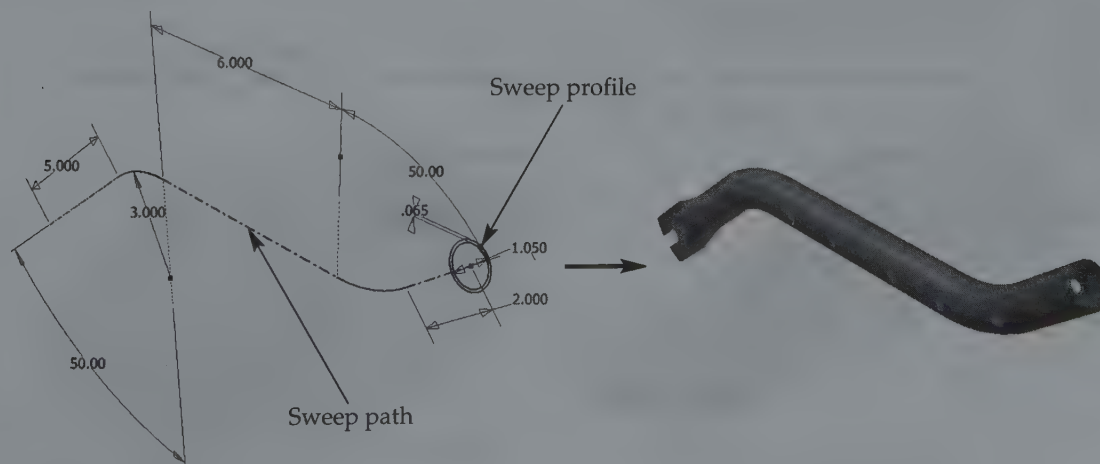
Figure 12-15.
Use the **Sweep** dialog box to create a sweep feature. Select an option from the **Type** area drop-down list to use a guide rail or surface.
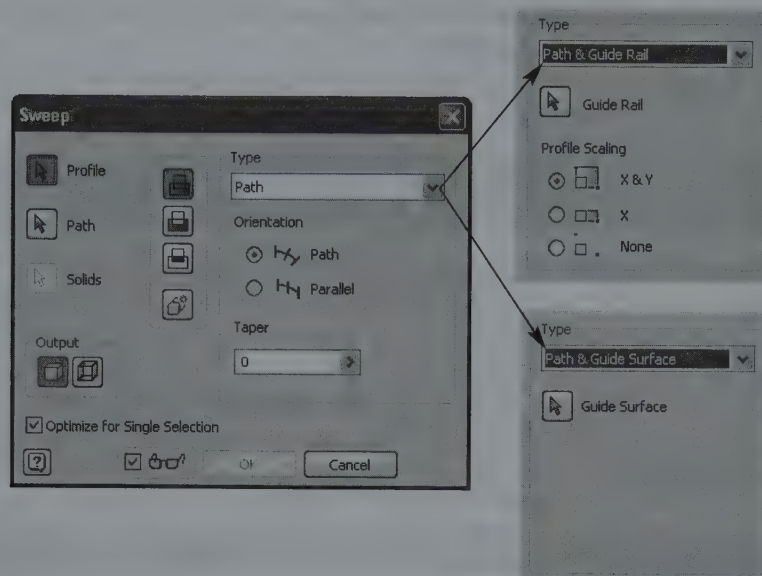
You have the option of sweeping a closed profile as a solid or surface by picking the appropriate button in the **Output** area. You can sweep an open profile only as a volumeless surface. When you are creating a solid base feature, the only possible operation is to form a new solid body, as evident by the selected **New Solid** button. When you add a sweep, the **Join**, **Cut**, and **Intersect** operation buttons are available for adding, removing, or combining material using the sweep.

Options in the **Type** area further define the sweep shape. The **Path** type is the default and creates a basic sweep feature based on a profile and path. Choose the **Path** radio button to produce a sweep that maintains a perpendicular constraint between the profile and path. Pick the **Parallel** radio button to create a sweep that becomes parallel to the profile when the path is not perpendicular to the profile. **Figure 12-16** shows the difference between the **Path** and **Parallel** options.

When using the **Path** option, you can specify a taper angle using the **Taper** text box. The taper angle defines the increasing or decreasing taper of the sweep profile as it follows the sweep path. The default angle of 0 applies no taper, resulting in a feature completely controlled by the sketch profile. A positive or negative taper angle constantly increases or decreases the size of the sweep profile. See **Figure 12-17**.

Figure 12-16.
The profile of a sweep feature created using the **Path** option maintains a perpendicular constraint between the profile and the path. The **Parallel** option of the **Path** type produces a different sweep, even with the same profile and path.
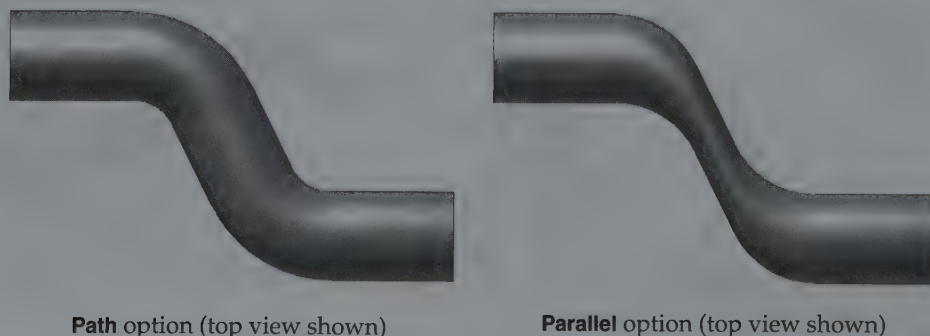
Path option (top view shown)   **Parallel** option (top view shown)

Figure 12-17.
These models compare the effects of positive and negative taper angles.

Start of sweep (profile)

End of sweep

Positive Taper Angle   Negative Taper Angle
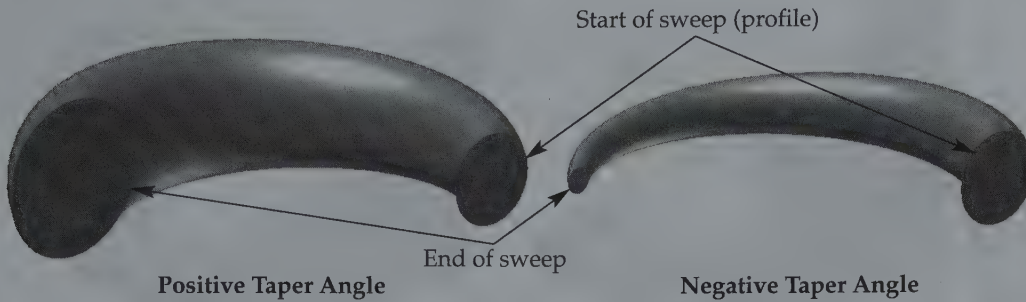
Guide Rail

Choose **Path & Guide Rail** type to include a *guide rail* with the sweep definition. Use the **Guide Rail** button to select a 2D or 3D sketch guide rail. Profile scaling controls the shape of a guide rail sweep. Pick the **X & Y** radio button to scale the sweep horizontally and vertically along the path. Choose the **X** radio button to scale the sweep horizontally along the path. Select the **None** radio button to eliminate scaling. **Figure 12-18** shows examples of each profile scaling option.

guide rail: A 2D or 3D sketched curve used with the sweep path to manipulate and further control the shape of a sweep.

Guide Surface

Pick **Path & Guide Surface** type to direct the form of a sweep along a *guide surface*. A guide surface is often required when a sweep profile follows a path along a complex surface. Use the **Guide Surface** button to select the surface. **Figure 12-19** shows an example of when a guide surface is required to produce the correct sweep shape. Pick the **OK** button to generate the sweep feature.

guide surface: A surface that helps control the shape of a sweep along a complex path.

Figure 12-18.
Using a guide rail to produce a sweep feature.

Path

Profile sketch   Guide rail

X and Y profile scaling

X profile scaling   No profile scaling
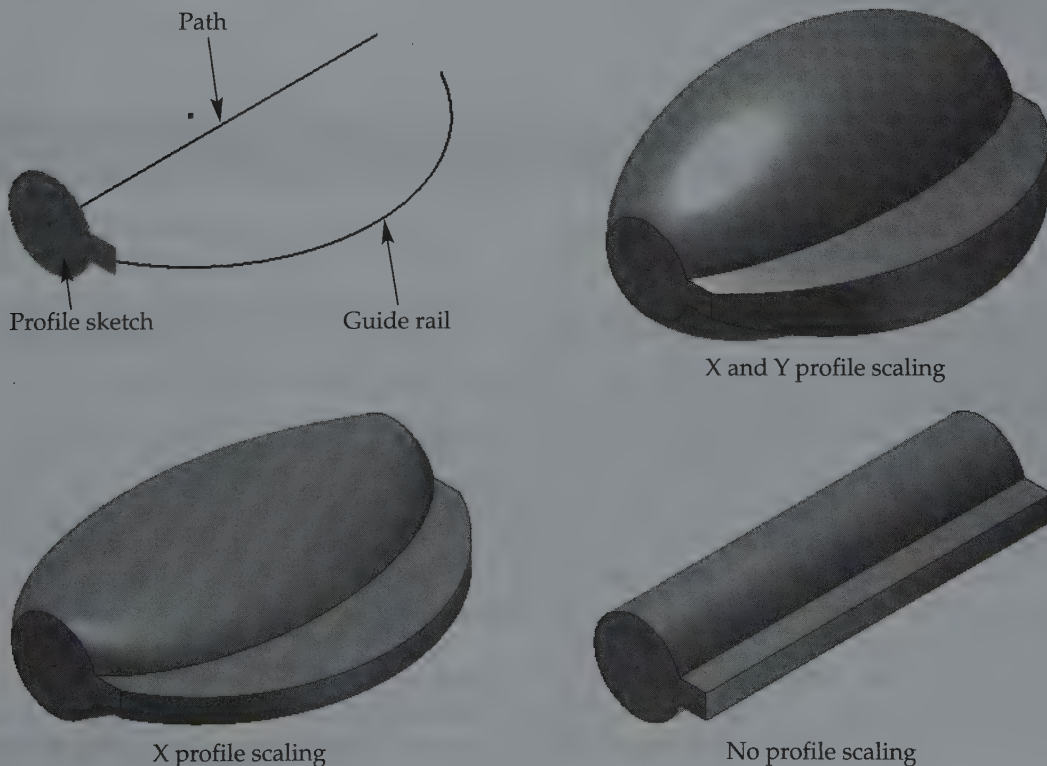
Figure 12-19.
Using a guide surface to direct the shape of a sweep.

Profile

Path

Path sweeps do not form according to the shape of an irregular surface

When a surface is selected, the sweep adjusts to, or in this example twists, according to the shape of the surface

Selected surface
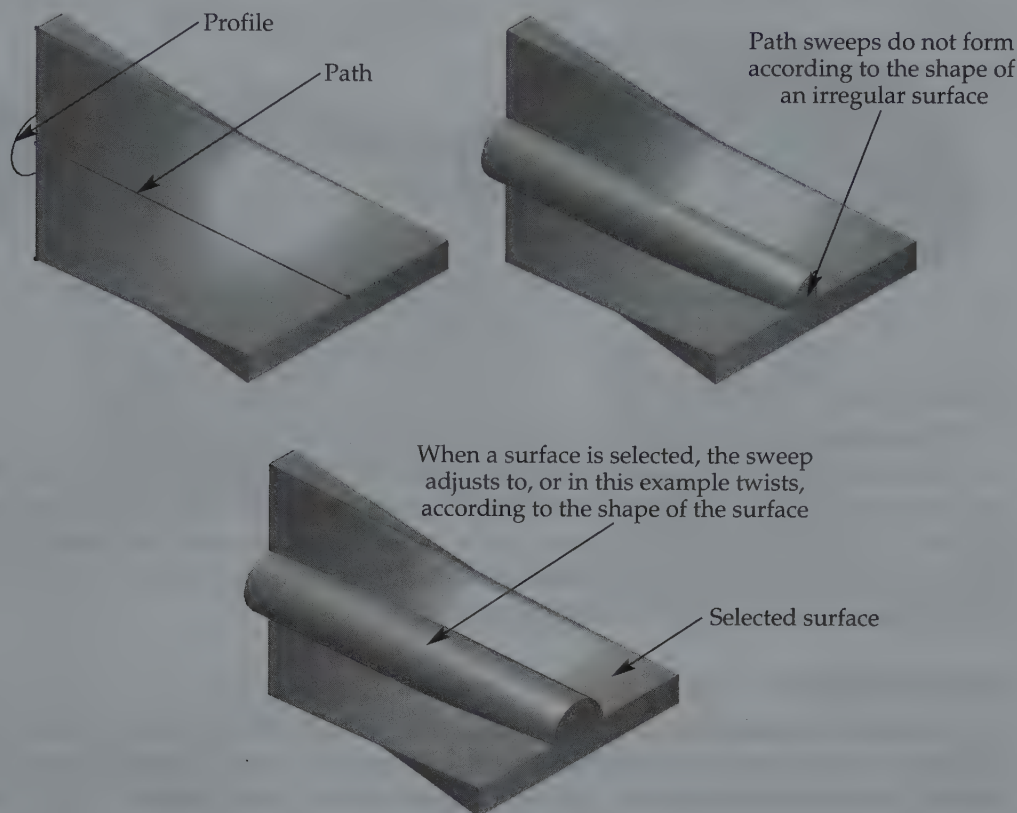
Exercise 12-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 12-5.

3D Sketch Features

3D sketches are required whenever it is not possible or is too difficult or time-consuming to build features using 2D sketches. 3D sketches significantly enhance your ability to produce features, including lofts and sweeps. Lofts use 3D rails and center-lines. Sweeps use 3D paths, rails, and guide surfaces. Sweeps that use 3D sketches are common for producing multi-axis parts and routing wires or pipes around existing components in an assembly. See **Figure 9-20.**

The main difference between lofting and sweeping with a 2D or 3D sketch is the process of creating sketches. The feature tools behave essentially the same. When preparing a loft, consider constructing 3D rails or a centerline first, so you can use the sketch geometry to position 2D sketch sections. Then access the **Loft** tool to generate the loft. When preparing a sweep, consider constructing a path, guide rail, and especially a guide surface before sketching the profile. Then access the **Sweep** tool to form the sweep.

Ribbon
Model
> Sketch

Create 3D Sketch
Shortcut
New 3D Sketch
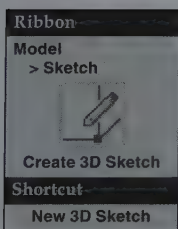
Figure 12-20.
A—A fishing hook built from a sweep with a 3D path.
B—A wire harness assembly with parts built using 3D sweeps.

A   B
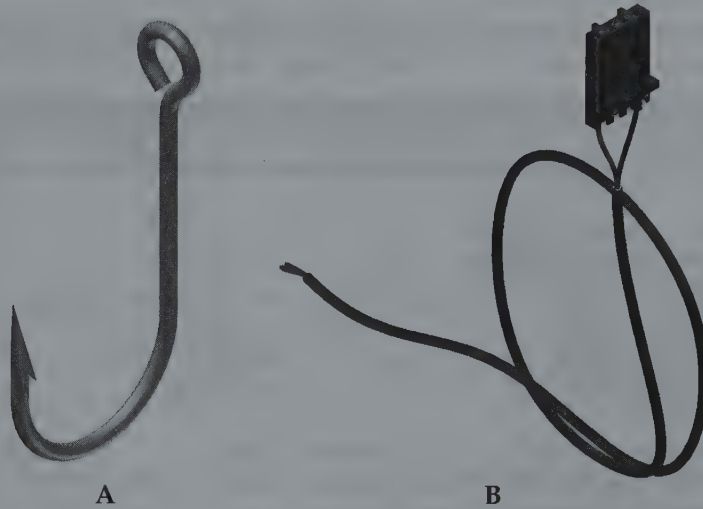
3D Sketching

Use the **3D Sketch** tool to begin a 3D sketch. The 3D sketch environment offers tools specifically for 3D sketching, but you will recognize many of the tools from the 2D sketch environment. The **Line**, **Spline**, **Three Point Arc**, **Center Point Arc**, and **Point, Center Point** tools are available for creating 2D and 3D sketches, but they behave differently in the 3D sketch environment because you are sketching in 3D space, not on a flat plane. Other tools, such as constraint and modification tools, act the same in 2D and 3D sketches. Remember, however, that when you use any tool in the 3D sketch environment, objects may not be on the same plane. Finish a 3D sketch as you would a 2D sketch.

Curves and Points

Access the **Line**, **Spline**, **Three Point Arc**, or **Center Point Arc** tool to create the corresponding 3D curve. Access the **Point, Center Point** tool to place sketch center points or sketch points. The **Inventor Precise Input** toolbar appears by default when you access any of these tools, allowing you to specify precise X, Y, and Z point coordinates. You can use the **Inventor Precise Input** toolbar or just pick points in space. Then add constraints as needed. Another option is to select existing points. All of these tools are available on the **3D Sketch** ribbon tab, which appears when you activate the **3D Sketch** tool.

> **PROFESSIONAL TIP**
>
> The **3D Snap Spacing** area on the **Modeling** tab of the **Document Settings** dialog box allows you to set default 3D snap settings. Enter a value in the **Distance Snap** text box to set the snap spacing for 3D sketching. Enter a value in the **Angle Snap** text box to set the snap spacing for angles when preparing 3D sketches.

Inventor Precise Input Toolbar

When you access 3D curve or point tools, the **Inventor Precise Input** toolbar and triad automatically appear. See **Figure 12-21**. You do not have to use the **Inventor Precise Input** toolbar or triad to locate points; you can just pick locations in space.

Figure 12-21.
The **Inventor Precise Input** toolbar and triad appear when you sketch a line or spline in the 3D sketch environment.

Reset
To Origin
button

Precise
Redefine
button

Precise
Delta
button

Initially located at the
center point (0,0,0)
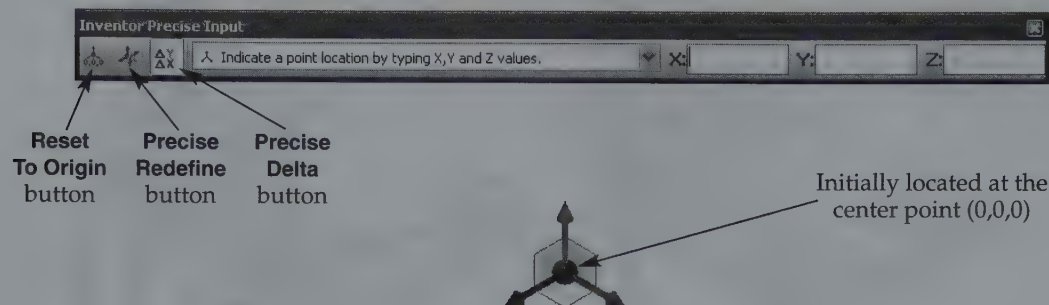
However, the **Inventor Precise Input** toolbar is effective for positioning points. Using the **Inventor Precise Input** toolbar involves entering values in the **X**, **Y**, and **Z** text boxes for each point. Type the value of the X coordinate in the **X** text box, the Y coordinate in the **Y** text box, and the Z coordinate in the **Z** text box, and then press [Enter] to locate the point.

The **Delta Input** button is active by default, allowing the triad to follow each point. The coordinates you enter are relative to the triad, or temporary origin. As a result, the first point of any new object is located at the temporary origin. For example, to sketch a line that begins at the default center point and ends at coordinates 1,3,3, enter 0 in the **X**, **Y**, and **Z** text boxes and press [Enter] to locate the first point. Next, type 1 in the **X** text box, 3 in the **Y** text box, and 3 in the **Z** text box, and press [Enter] to locate the second point. To continue the line to end at 3,5,5 in reference to the model origin, type 2 in the **X** text box, 2 in the **Y** text box, and 2 in the **Z** text box, and press [Enter] to locate the third point. Press [Esc], right-click and pick **Done**, or access another tool to exit.

Use the **Precise Redefine** button to adjust the position or orientation of the triad. Pick the triad sphere followed by a point to relocate the triad origin. Pick a triad axis followed by a line, curve, or point to align the axis with the selection. Pick the **Reset to Origin** button to return the triad to the model origin, allowing you to specify coordinates in reference to the origin instead of the triad.

NOTE

You can select a triad plane to add a 0 to the X, Y, or Z text box and define the location of a point according to the selected plane.

Referencing Points

Pick work points, sketch points, or linear feature edge endpoints and midpoints to locate 3D points. You can also pick a circular feature edge to locate a point at the center of the edge. This is an effective way to reference the center of a circular edge, such as a cylinder, without adding a point or creating a sketch. Pick points in sequential order. A coincident constraint links the curve or point with the selection. Press [Esc], right-click and pick **Done**, or access another tool to exit. **Figure 12-22** shows an example of picking work points to create a 3D sweep path using 3D lines. You can accomplish the sweep path using the **Inventor Precise Input** toolbar and triad. **Figure 12-23** shows an example of picking feature edge endpoints and midpoints to create a 3D sweep path using a continuous closed tangent spline.

Figure 12-22.
An example of picking work points to position the ends of 3D lines to use as a sweep path. You can use the **Inventor Precise Input** toolbar and triad to sketch the same path without creating work points.

Work Points Used to Create 3D Sketch

Sweep Sketches

Completed Sweep
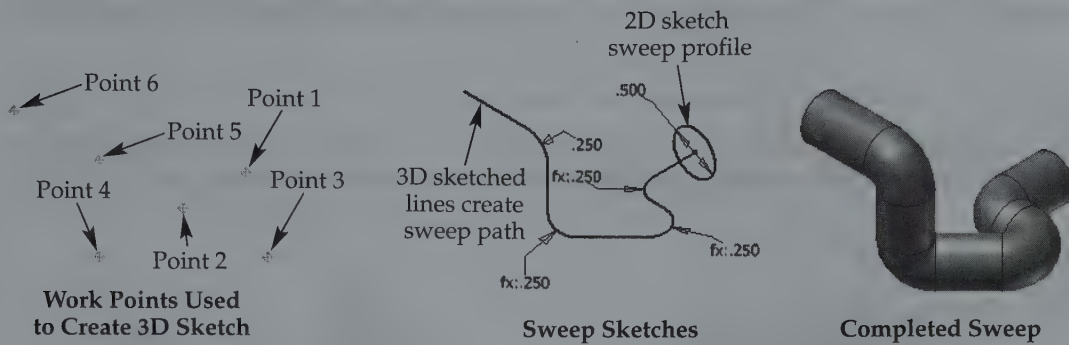
Figure 12-23.
An example of picking existing feature endpoints and midpoints to position the points of a 3D spline to use as a sweep path. This sweep uses a cut operation. Notice the rounds added before the sweep operation.

3D Sketch

Sweep Sketches

Completed Sweep
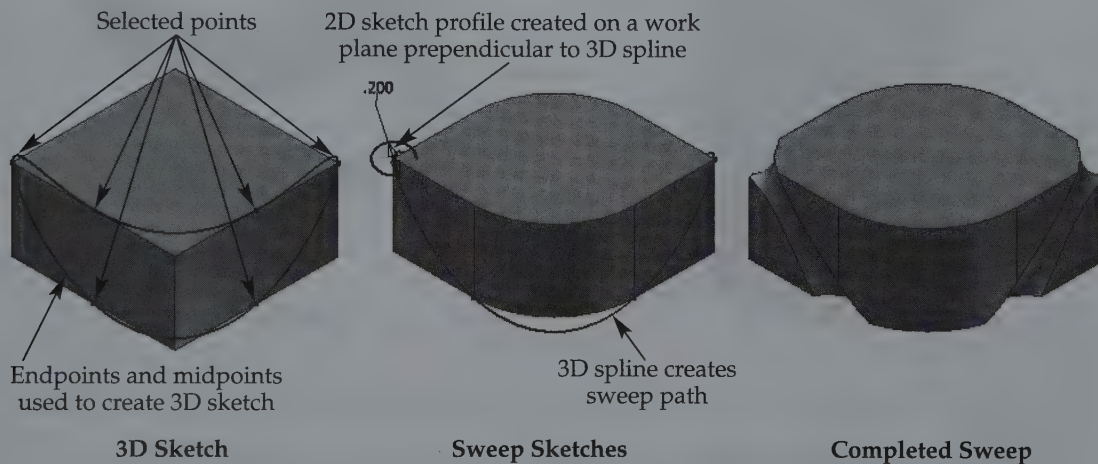
Helical Curves

Use the **Helical Curve** tool to create a *sketch helix*. Sketch helixes are most commonly used to form loft rails or a centerline, or a sweep path or guide rail. The **Coil** tool creates similar features, without an extra 3D sketch, but limits your ability to construct certain shapes, such as the loft shown in **Figure 12-24**.

> **sketch helix:** A winding spiral shape primarily used to create springs, detailed threads, and similar items.

PROFESSIONAL TIP

Use a helical curve to create a sweep in which the sweep profile spirals along the helix path, perpendicular to the helix. To create a sketch profile perpendicular to the helix curve, sketch on a work plane positioned by picking the helix and the helix start point or endpoint.

Ribbon

3D Sketch > Draw
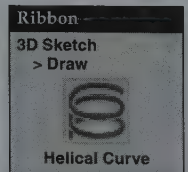
Helical Curve

Access the **Helical Curve** tool to display the **Helical Curve** dialog box and the **Inventor Precise Input** toolbar and triad. See **Figure 12-25**. The first phase of helix sketching involves specifying the start point of the helix axis, followed by the axis endpoint, and finally the helix start point, where the spiral begins. Use the **Inventor Precise Input** toolbar, pick points in space, or pick existing sketch or feature points. See

Figure 12-26. The helix values you specify, or those that Inventor calculates, such as height and diameter, ultimately define the size and location of the helix.

After you locate helix points, use the options in the **Helix Shape** and **Helix Ends** tabs of the **Helical Curve** dialog box to adjust the helix shape and ends. The **Helical Curve** dialog box includes essentially the same options as the **Coil** dialog box, with the

---

Figure 12-24.
A loft created using a sketched helical curve as a centerline. The sections in this example are sketches on work planes created by picking the helix endpoint and the helix curve.

Loft Sketches   Completed Loft
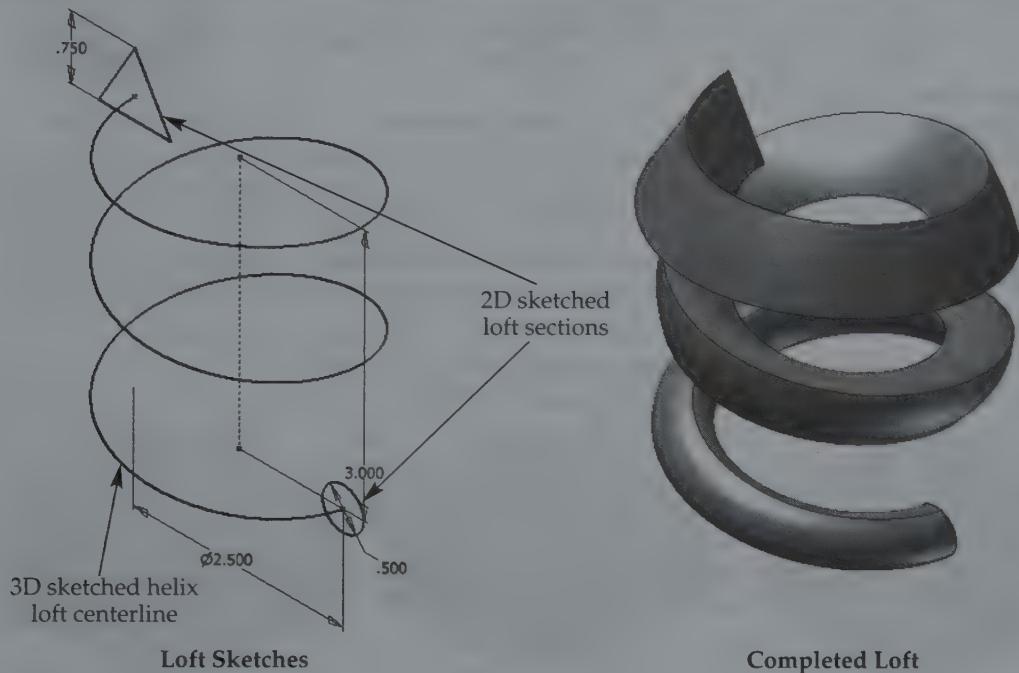
Figure 12-25.
The **Helical Curve** dialog box, **Inventor Precise Input** toolbar, and triad appear when you access the **Helical Curve** tool.
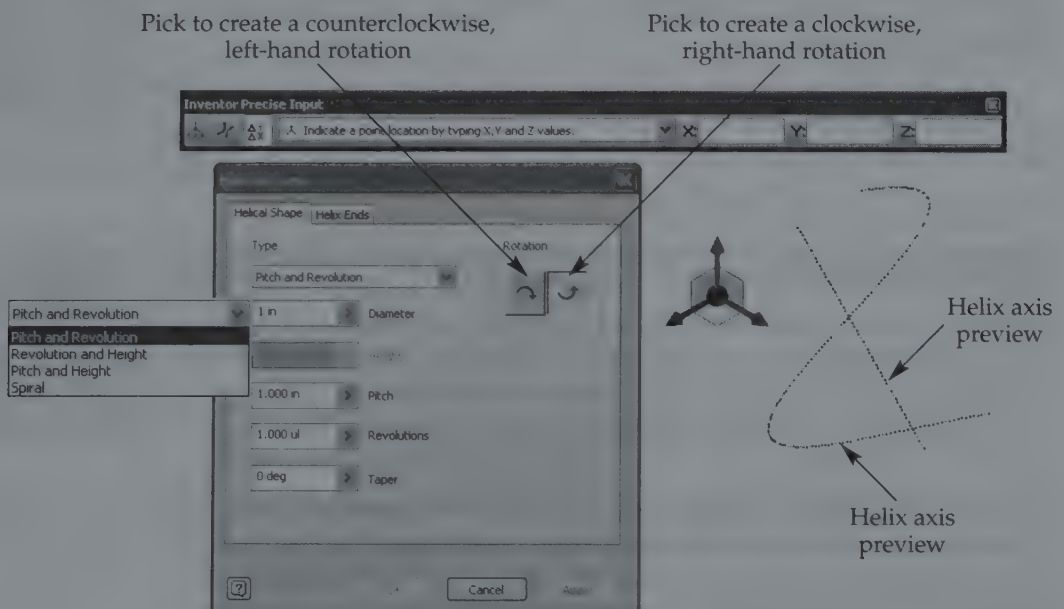
addition of the **Diameter** setting in the **Helix Shape** tab. The **Spiral** helical shape type, which is similar to the **Spiral** option of the **Coil** tool, allows you to create a helical shape without any height or optional taper. See **Figure 12-27**. Pick the **Apply** button to create the helix. Continue forming helixes, or pick the **OK** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

Figure 12-26.
A—An example of a helix placed using coordinates entered in the **Inventor Precise Input** toolbar. B—A common application of picking cylinder edges to place the helix axis through the center of a cylinder.
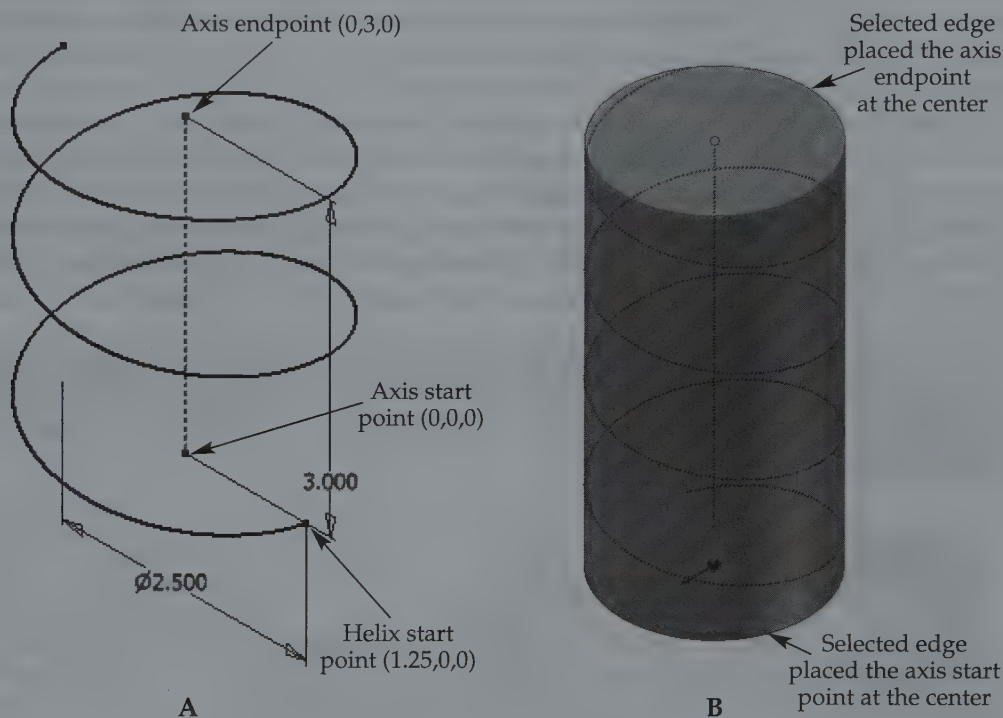
Figure 12-27.
A loft created using a sketched spiral helical curve as a centerline. The sections in this example are sketches on work planes created by picking the helix endpoint and the helix curve.

Exercise 12-6

Bends

Ribbon
3D Sketch > Draw
Bend
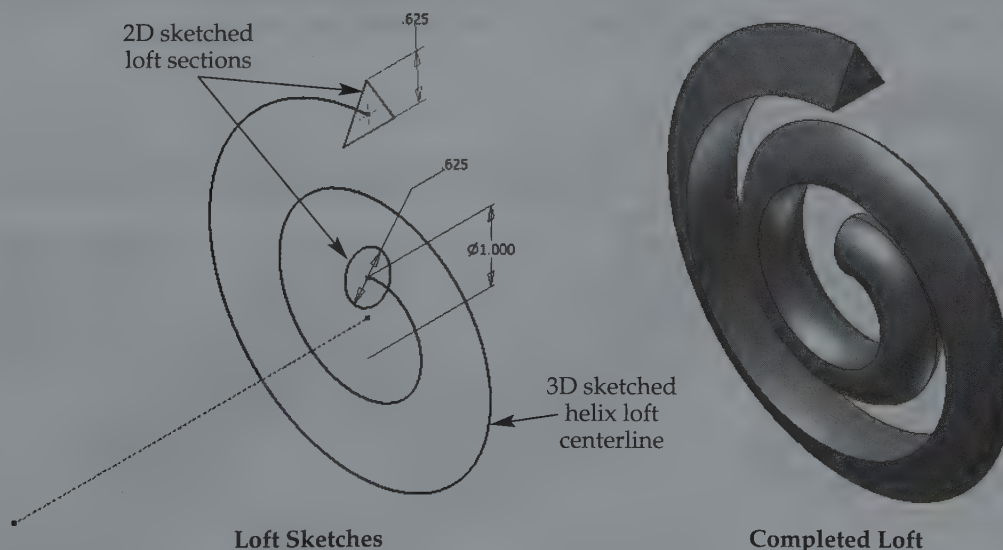
Some applications, most often sweep paths, require a radius, or corner bend, at the intersection of 3D curves. Access the **Bend** tool to add a corner bend using the **Bend** dialog box. See **Figure 12-28**. Specify a bend radius using the text box, and if necessary, select the **Equal** button to create bends with equal radii without adding additional dimensional constraints. Then select the intersecting corners of curves or the two curves to bend.

You have the option of automatically creating corner bends while sketching connected lines by picking the **Auto-Bend with 3D Line Creation** check box in the **Sketch** tab of the **Application Options** dialog box. Then use the **Auto-Bend Radius** text box in the **Sketch** tab of the **Document Settings** dialog box to set an automatic bend radius. Deselect the **Auto-Bend with 3D Line Creation** check box, or access the **Line** tool, right-click, and pick **Auto-Bend** to deactivate automatic corner bending.

> **NOTE**
>
> Bend options do not apply to splines, but spline adjustment options are available.

Exercise 12-7

Including Geometry

Ribbon
3D Sketch > Draw
Include Geometry
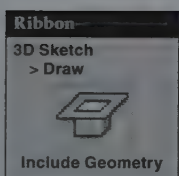
Converting 2D sketch geometry or model elements into 3D sketch geometry is one of the easiest methods of preparing a 3D sketch. Access the **Include Geometry** tool and pick items to add to the 3D sketch. You can select 2D sketch shapes or feature edges. Press [Esc], right-click and pick **Done**, or access another tool to exit.

Exercise 12-8

Figure 12-28.
Creating a 3D sketch bend using the **Bend** dialog box.

Bend

0.250 in

Select the corner or individual lines to add a bend
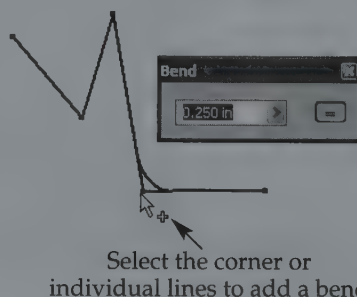
Intersection Curves

Access the **3D Intersection Curve** tool to place a 3D curve at the intersection of two surfaces, planes, or faces using the **3D Intersection Curve** dialog box. See **Figure 12-29**. Pick the first surface, plane, or feature face, followed by the second surface, plane, or feature face. To redefine the first or second selection, pick the **1** or **2** button and then choose the new surface. Pick the **OK** button to generate the sketch shape.
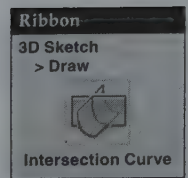
Silhouette Curves

The **Silhouette Curve** tool constructs a curve around the widest geometry of a solid body, according to a specified direction. Depending on the model, the curve may form at a point of tangency. **Figure 12-30** shows an example of using the **Silhouette Curve** tool to form a sweep path. Another common application is developing a *parting line* for molded product design. The **Silhouette Curve** tool calculates the widest point on the part, from which draft always falls, making the tool especially effective for situations when you cannot use a plane to specify the parting line or surfaces.

The **Create Silhouette Curve** dialog box appears when you access the **Silhouette Curve** tool. If the model contains a single solid body, the body is selected by default. Otherwise, use the **Body** button to select a body to assign the curve. Then use the **Direction** button to specify where the curve forms. In molded product design, the direction is the *pull direction*. Pick a face or plane perpendicular to the direction, or select a line, edge, or axis collinear to the direction. Pick the **Apply** button to create the curve. Continue forming curves, or pick the **OK** button, press [Esc], right-click and pick **Done**, or access another tool to exit.
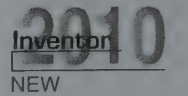
parting line: The line that defines the surfaces where two or more molds or dies meet during a molding process.

pull direction: The direction in which a casting mold is pulled or removed from the part.

Figure 12-29.
Creating a 3D sketch using existing surfaces, planes, and faces and the **3D Intersection Curve** dialog box.
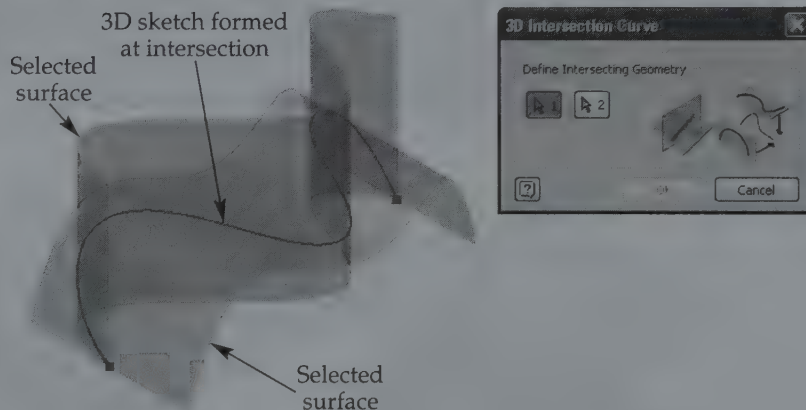
Figure 12-30.
The **Silhouette Curve** tool is effective for constructing a curve at the widest points of a model. The example shown is a sweep path, but it could easily be a parting line for a molded product design.
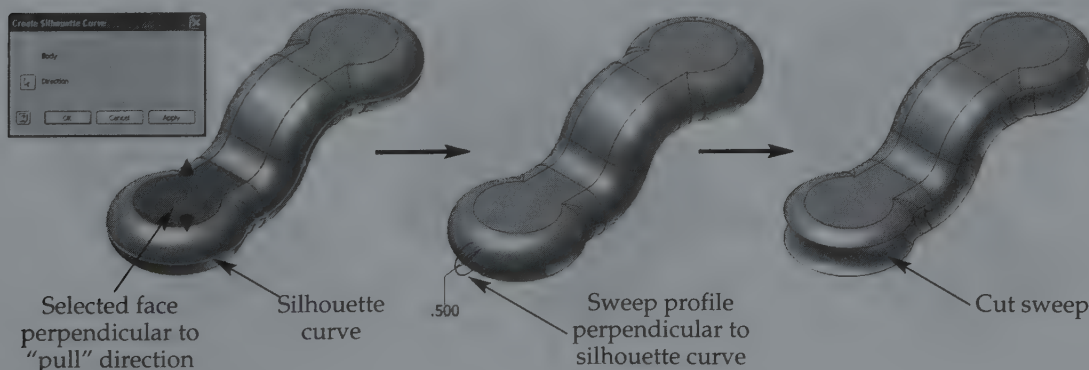
The **Boundary Path** and **Split** tools, described in Chapter 13, are often used with the **Silhouette Curve** tool to design molded components.

Exercise 12-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 12-9.

Curve Projection

Access the **Project Curve to Surface** tool to project a curve onto a surface to create a 3D sketch profile using the **Project Curve to Surface** dialog box. See **Figure 12-31**. Use the **Faces** button to choose the surface faces on which to project the curve. Next, pick the

Figure 12-31.
A—The **Project Curve to Surface** dialog box. B—Projecting a curve onto a surface using the **Project Along Vector** output method.

Pick to project the curve to the surface using a vector direction

Pick to wrap the curve along and around the selected surface

Pick to project the curve to the surface according to the closest distance between the curve and the surface

2D sketched curve

Face

Vector direction

Projected curve

Loft created using 2D and 3D sketch sections

A

B
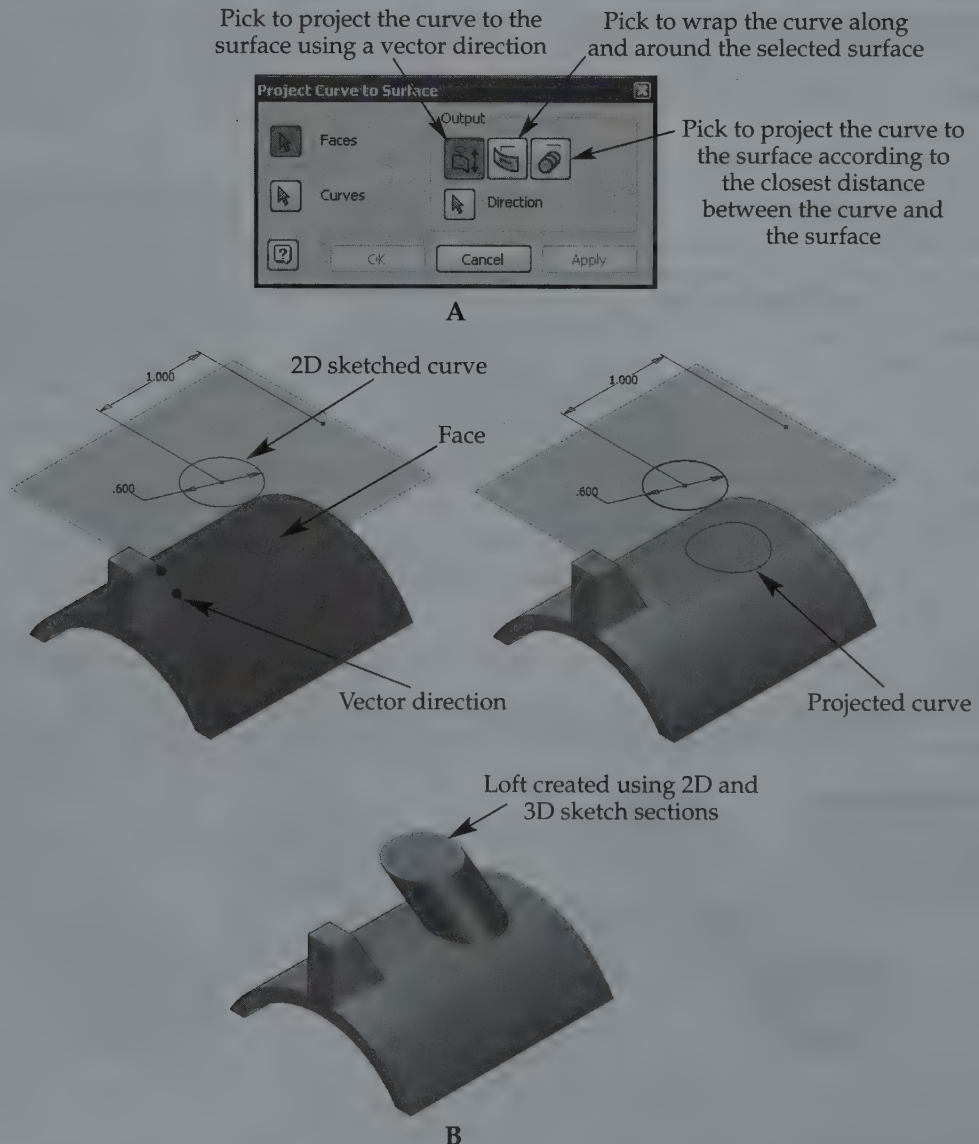
Curves button and select the curves to project. A curve is usually a 2D sketch shape. You can select multiple curve profiles, as long as they are on the same sketch plane.

The projection option shown in **Figure 12-31** uses the **Project Along Vector** button to project the curve onto the surface using a vector direction. Choose the **Direction** button and select the vector direction, which can be an existing linear feature edge or a work axis. Pick the **Apply** button to project the curve and remain in the **Project Curve to Surface** tool, or select the **OK** button to project the curve and exit.

The projection option shown in **Figure 12-32** uses the **Wrap to Surface** button to wrap the curve along and around the surface based on the tangent relationship of the surface and the curve. No additional selection is necessary. Pick the **Apply** button to project the curve and remain in the **Project Curve to Surface** tool, or select the **OK** button to project the curve and exit.

In **Figure 12-33**, the **Project to Closest Point** button is used to project the curve onto the surface according to the closest distance from the curve to the surface. No additional selection is necessary. Pick the **Apply** button to project the curve and remain in the **Project Curve to Surface** tool, or select the **OK** button to project the curve and exit.

Ribbon
3D Sketch
> Draw

**Project Curve
to Surface**
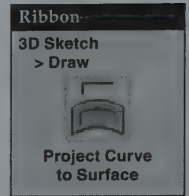
Figure 12-32.
Projecting a curve onto a surface using the **Wrap to Surface** output method.
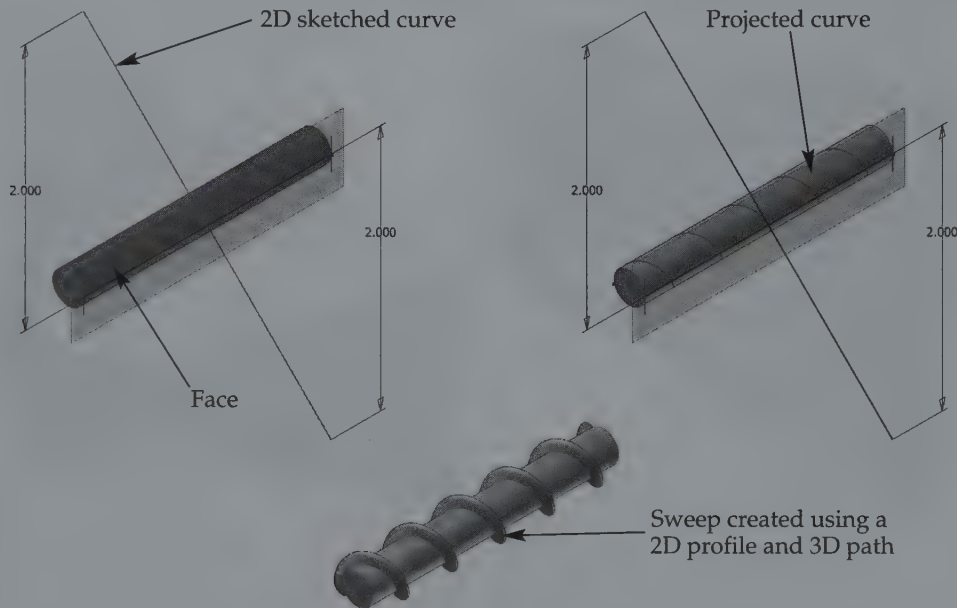
Figure 12-33.
Projecting a curve onto a surface using the **Project to Closest Point** output method.
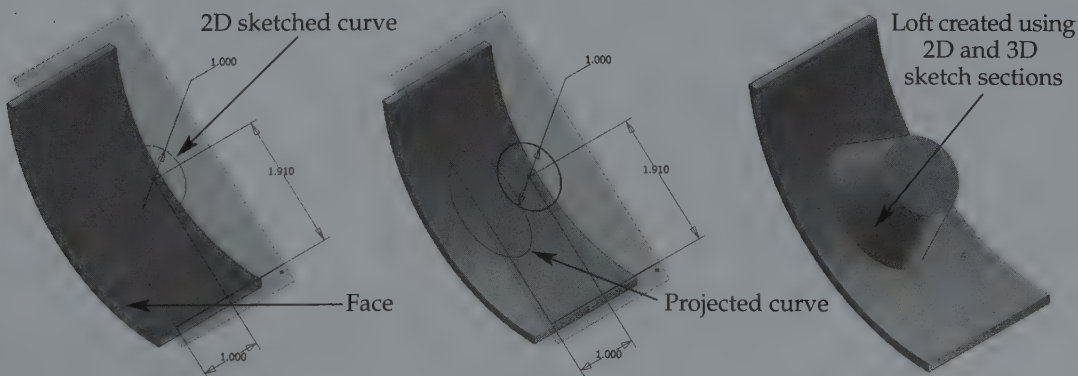
Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. Define *loft*.
2. What are sections?
3. Describe a closed loop control and specify when it is available.
4. Briefly describe a rail.
5. Give the basic function of a loft centerline.
6. What are placed sections and how are they calculated?
7. Define *sweep*.
8. Identify two characteristics required to create a sweep?
9. What must occur to be sure that a sweep generates without errors?
10. Briefly describe a guide rail.
11. What is a guide surface?
12. Describe the basic function of a sketch helix.
13. What is the purpose of the **Silhouette Curve** tool?
14. What is a parting line?
15. In molded part design, what is the pull direction?

Problems

- *Open the specified part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **File:** EX12-1.ipt
   Save as: P12-1.ipt
   Title: NOZZLE
   Specific Instructions:
   A. Open a 2D sketch on Face1 and offset the projected edges 50 mm.
   B. Open a 2D sketch on Face2 and offset the projected edges 15 mm.
   C. Use the **Loft** tool to create a loft. Use a cut operation and specify Sketch3 and Sketch4 as sections. Apply a free condition for the Sketch3 section. Use a direction condition for the Sketch4 section with a 90°angle and 100 ul weight. Leave all other values set as default.
   D. Add all fillets and rounds using a radius of 5 mm. The final part should look like the part shown on the right.

Face1
(not shown)

Face2

Face2
(not shown)

Face1
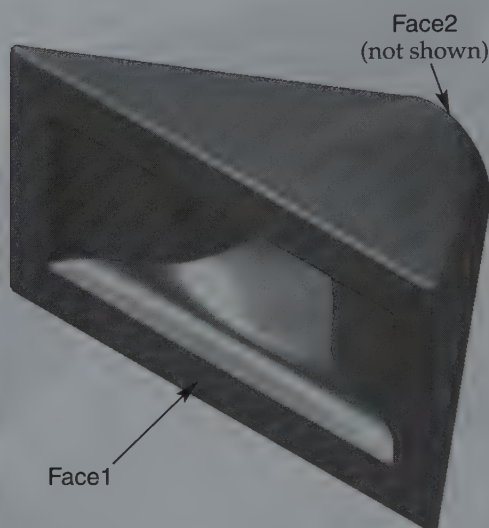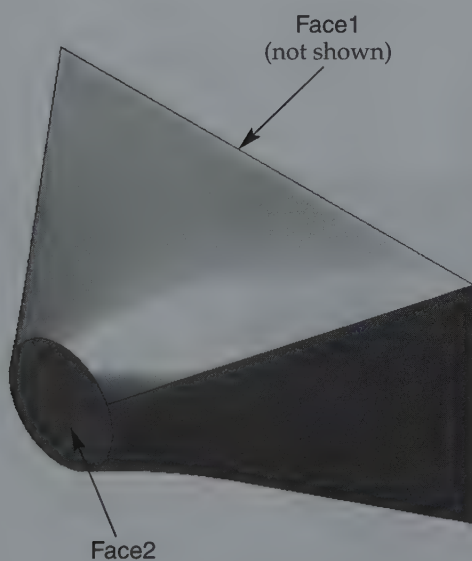
▼ Basic

2. **File:** EX12-7.ipt
   Save as: P12-2.ipt
   Title: HELIX
   Specific Instructions: Open a 2D sketch on the face shown and sketch the geometry shown. Make the 3D sketch of the helix visible for additional use. Sweep the new 2D sketch profile along the visible 3D sketched helix path. Use a cut operation and path type and orientation, and leave all other values set as default. Turn off the visibility of the 3D sketch. Edit the 2D sketch of the triangle, now consumed by the loft, to change the size of the triangle from 1.000 to .875. The final part should look like the part shown.

Open a sketch on this face

.500

Project this face to project the circular edge center point

Center of polygon coincident to projected edge center point

Horizontal

.500
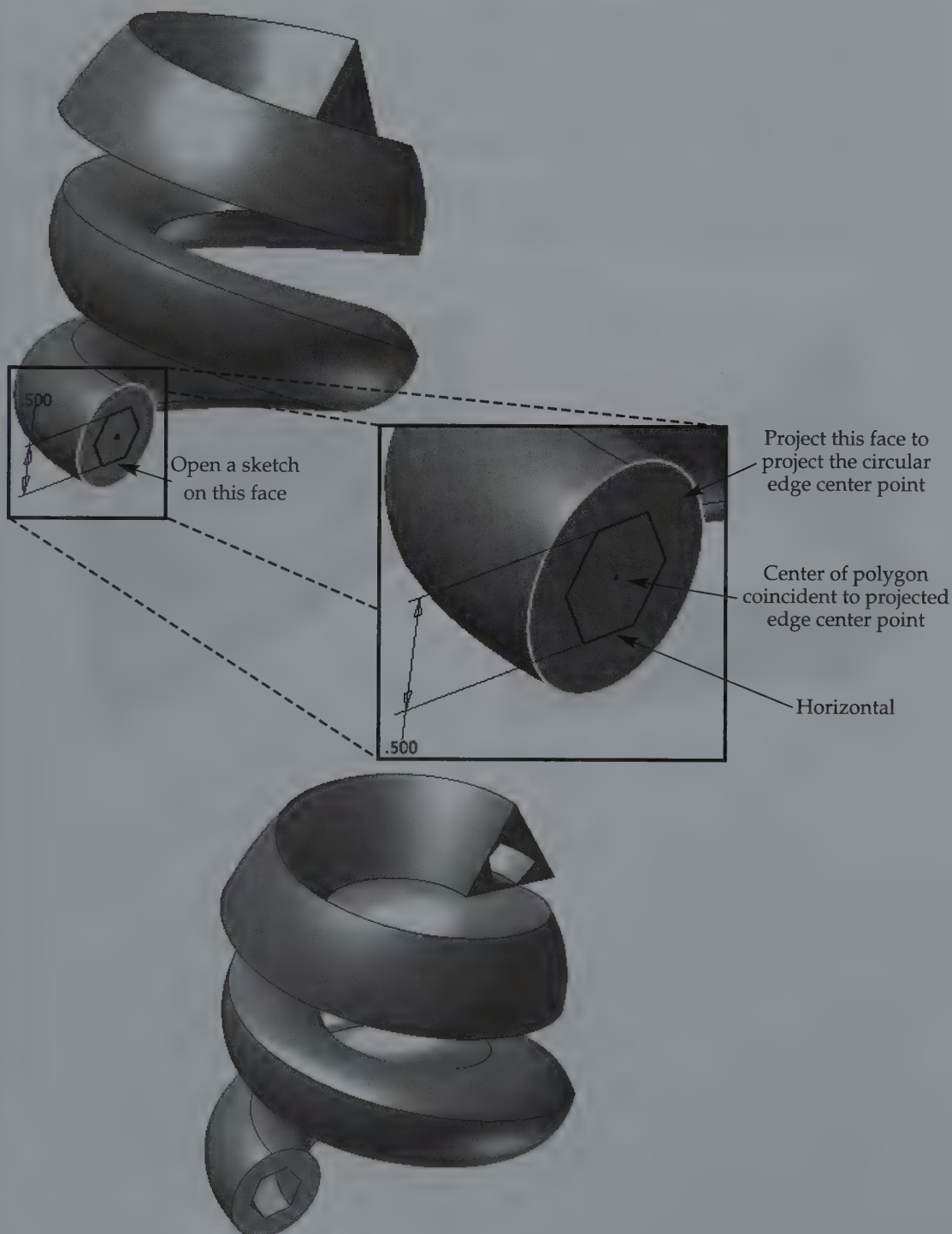
3. **File:** P7-3.ipt
   Save as: P12-3.ipt
   Title: STOP
   Specific Instructions: Create the 2D sweep profile sketch on the face shown. Create the 2D sweep path sketch on the face shown. Cut sweep the profile along the path as shown. Add the specified rounds. To add the knurled representation, hold down the [Ctrl] key and select the faces shown, seven total. Then right-click, pick **Properties**, and select the Metal-Steel (Knurled) face color style. Open a sketch on the XY plane and sketch the geometry shown. Slice the graphics to see the sketch plane. Cut-extrude the sketch .065" midplane.

Sketch these short lines

Profile sketch plane

Tangent

Vertical

1.350

.610

Concentric

Project

.300

Line collinear with edge

Full round radius

Loop select, .01 radius

Full round radius

Projected points

.160

.100

4–8 Instructions:

- *Create sketches of the following objects*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

4. **Title:** 90° REDUCING ELBOW
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-024-01
   Project: SPOUT
   Material: Brass, Soft Yellow
   Color: Nickel (Bright)
   Save as: P12-4.ipt
   Specific Instructions: Sketch a Ø2″ circle around the projected center point on the XZ plane. Open a new sketch on the XY plane and sketch the arc shown. Use the **Sweep** tool to create a solid sweep feature and −6° taper angle. Leave all other values set as default. Shell the sweep feature to create .125″ thick walls, and remove the end faces. Add .1″ chamfers to the end edges. The final part should look like the part shown.
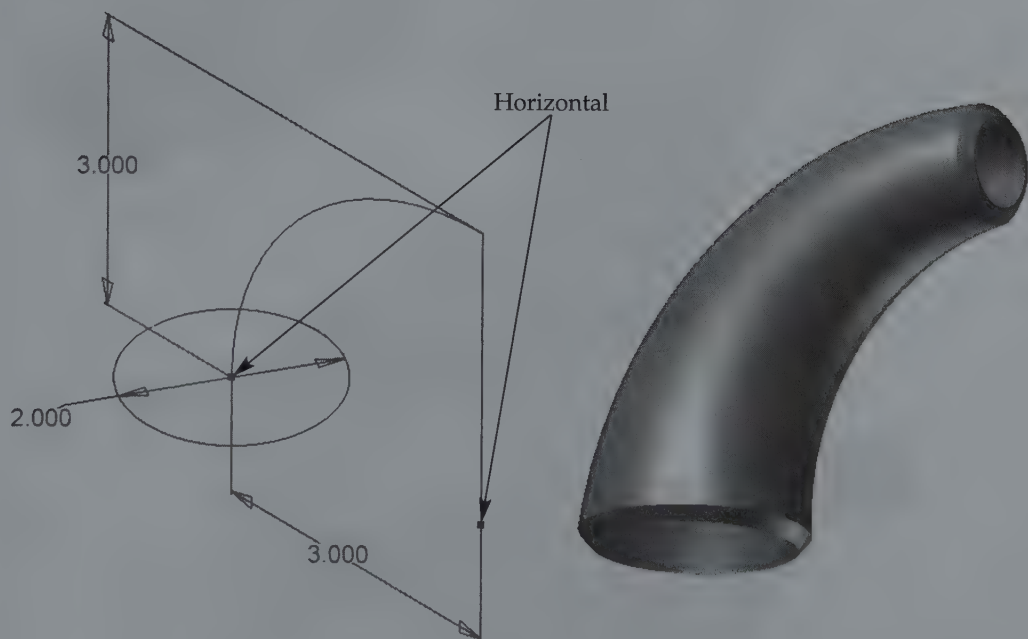
5. **Title:** FORK BODY
   Units: Metric
   Template: Part-mm.ipt
   Part Number: IAA-025-01
   Project: FORK
   Material: Stainless Steel
   Color: As Material
   Save as: P12-5.ipt

Specific Instructions: Sketch a 4 mm × 4 mm rectangle centered around the projected center point on the YZ plane. Open a new sketch on the XY plane, and sketch the geometry shown. Use the **Sweep** tool to create a solid sweep feature using Sketch1 as the profile and Sketch2 as the path. Leave all values set as default. Open a sketch on Face1, and project the edges. Extrude Sketch3 150 mm in the positive direction. Place 2.5 mm × 15 mm chamfers on the front edges of the fork, as shown. Add all fillets using a radius of 2 mm. Add all rounds using a radius of 0.5 mm. The final part should look like the part shown.

15

60

Projected
center point

10

5

Chamfers
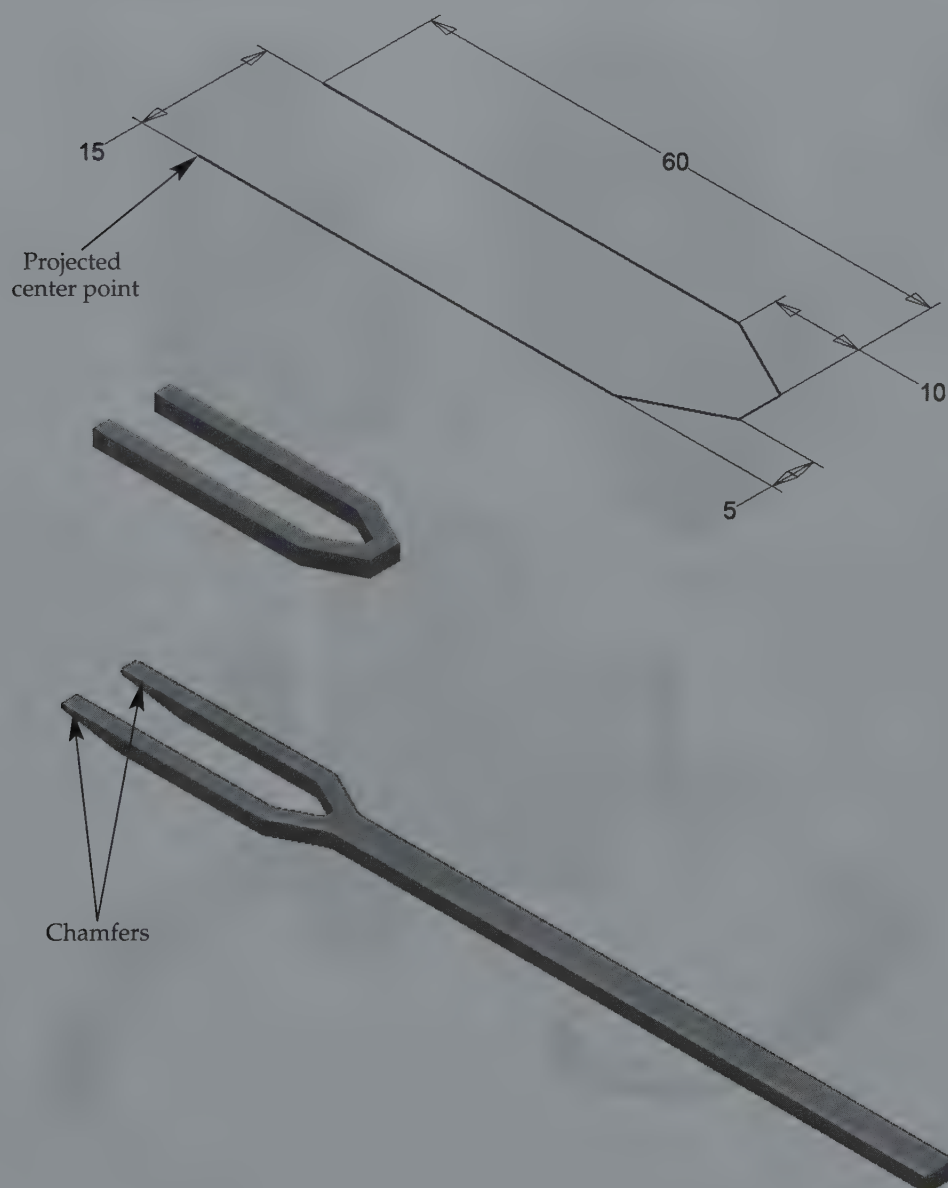
6. **Title:** HANDLE
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-026-01
   Project: HARDWARE
   Material: Default
   Color: Default
   Save as: P12-6.ipt
   Specific Instructions: Create the sketch shown on the XY plane. Extrude the sketch (two profiles) .5" in the positive direction. Offset a work plane 1.75" from the YZ plane. Sketch the geometry shown on the work plane. Use the **Loft** tool to create the loft feature shown by selecting Face1, followed by Sketch2, followed by Face2 as the loft sections. Apply a join operation, and leave all other values set as default. Place the specified holes in the centers of the extrusion bases, as shown. The final part should look like the part shown.
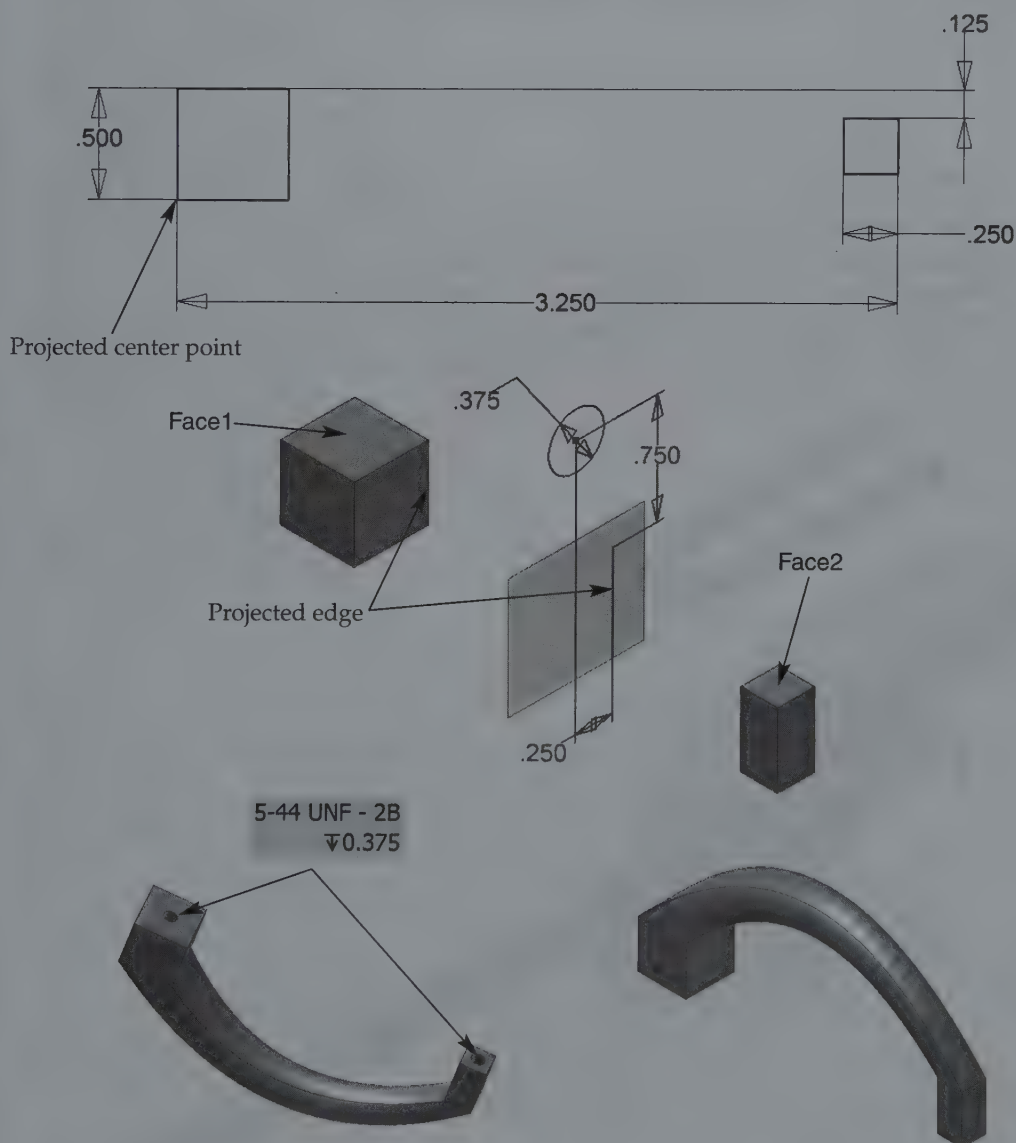
7. **Title:** WIRE
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-027-01
   Project: WIRE
   Material: Default
   Color: Default
   Save as: P12-7.ipt

   Specific Instructions: Create an extrusion similar to the extrusion shown. Using the 3D sweep techniques described and the following information, create a wire similar to the wire shown. Open a 2D sketch and sketch a circle for the wire profile. Finish the 2D sketch. Open a 3D sketch, and offset the required work planes. Use the work planes to place work points, similar to the work planes and points shown. Connect the work points with a line and bends. This creates the sweep path. Finish the 3D sketch. Access the **Sweep** tool and generate the sweep feature. The final part should look similar to the part shown.

Problems - Chapter 12
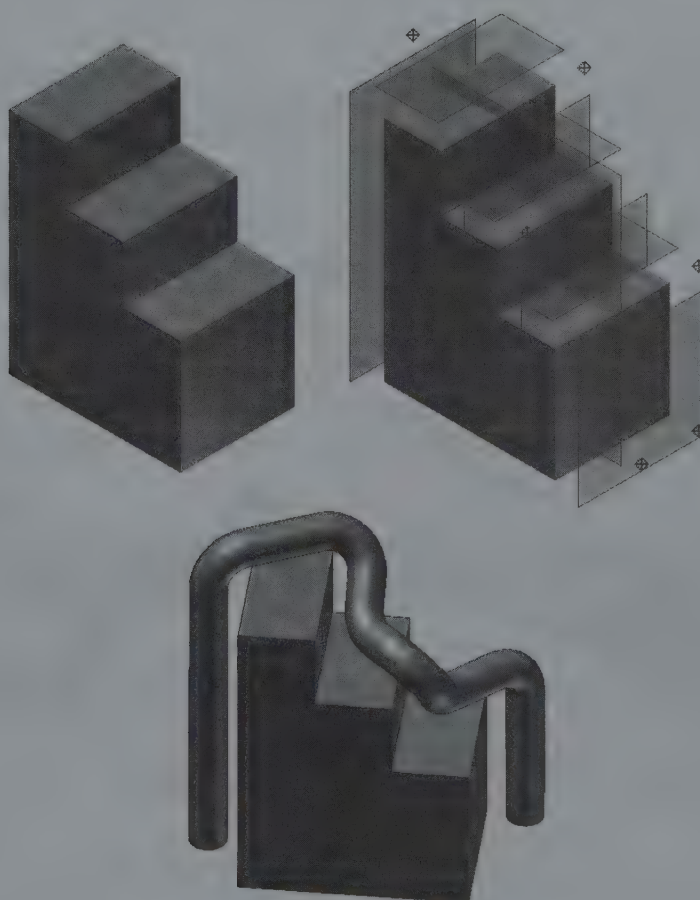
8. **Title:** TILLER BLADE
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-028-01
   Project: TILLER
   Material: Choose an appropriate material
   Color: Choose an appropriate color
   Save as: P12-8.ipt
   Specific Instructions: Create a tiller blade similar to the one shown according to the following information. Create four equally spaced work planes. Open sketches on the work planes and sketch rectangles at different angles. Loft the sketches using manual point mapping. Round the sharp end corners using the **Fillet** tool. Extrude a center support. Place a hole in the center of the center support. Use a circular pattern for the loft and fillet features. The final part should look similar to the part shown.

Projected center point

1

2

3

4
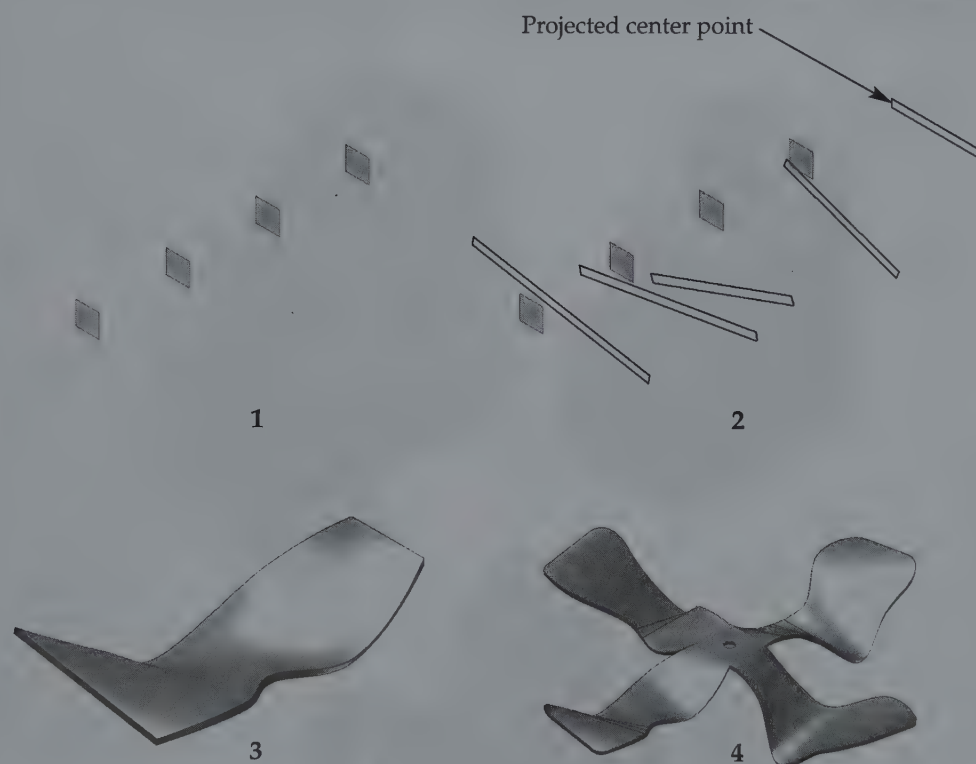
Split, Emboss, Decal, and Surface Tools

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create split features.
✓ Emboss and engrave.
✓ Add decals.
✓ Use additional surface tools and techniques.

Add splits, embossing, engraving, and decals to replicate specific manufacturing or modeling processes. You can also use these features, as you do others, in a more general way to develop a part model. This chapter also explores additional surface tools and techniques that expand your ability to create models.

Splits

A *split* feature is common in molded part design, but it is also suitable for many other modeling requirements. Before you can apply a split, you must establish or identify the *split tool*. One option is to reference a plane in the **Origin** folder of the browser. A split tool can also be a work plane, 2D or 3D sketch, or construction surface. **Figure 13-1** shows examples of split tools. Access the **Split** tool to create a split using the **Split** dialog box. See **Figure 13-2**.

Splitting Faces

The **Split Face** button is active by default, allowing you to split feature faces without removing material or forming separate solid bodies. See **Figure 13-2**. The **Split Face** function is useful for many applications, such as applying draft, thickening, and offsetting. Use the **Split Tool** button to select the split tool. Select the **Faces to Split** button to split specific faces, and select the faces using the **Faces** button. Choose the **All** button to split all faces intersecting the split tool, and use the **Faces** button to select a solid body. Pick the **Apply** button to create the split. Continue splitting, or pick the **OK** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

Ribbon

Model > Modify

Split

split: A feature that removes a portion of a model or divides faces at a separation sketch or plane.

split tool: The parting line, plane, or surface where separation occurs when you create a split feature.

Figure 13-1.
If an origin plane is not appropriate, a split tool, or split separation definition, can be a work plane, sketch, or surface.

Work Plane (Offset)   Sketch (2D Line)   Surface (Extrusion)
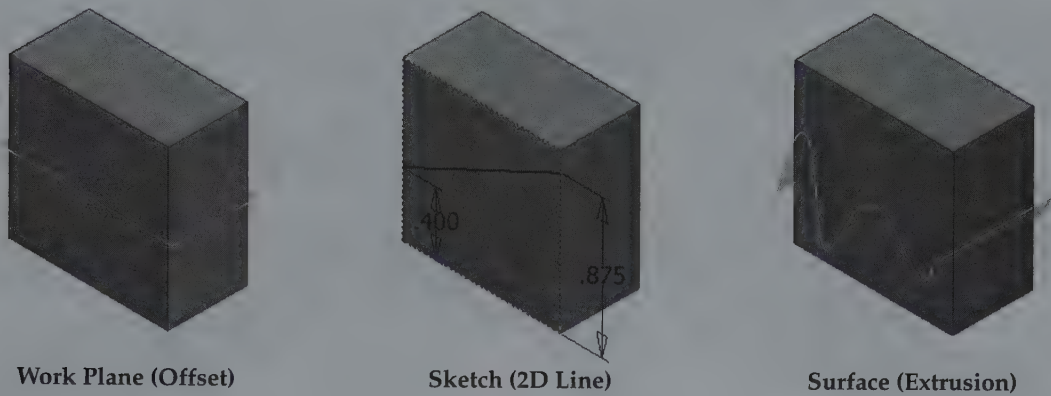
Figure 13-2.
Using the **Split Face** method in the **Split** dialog box to split a face. Draft is a common application for split faces.

Split Face button

Trim Solid button

Split Solid button

Pick to split all faces

Selected face

Face Draft tool applies draft

Split tool (2D sketched arc)
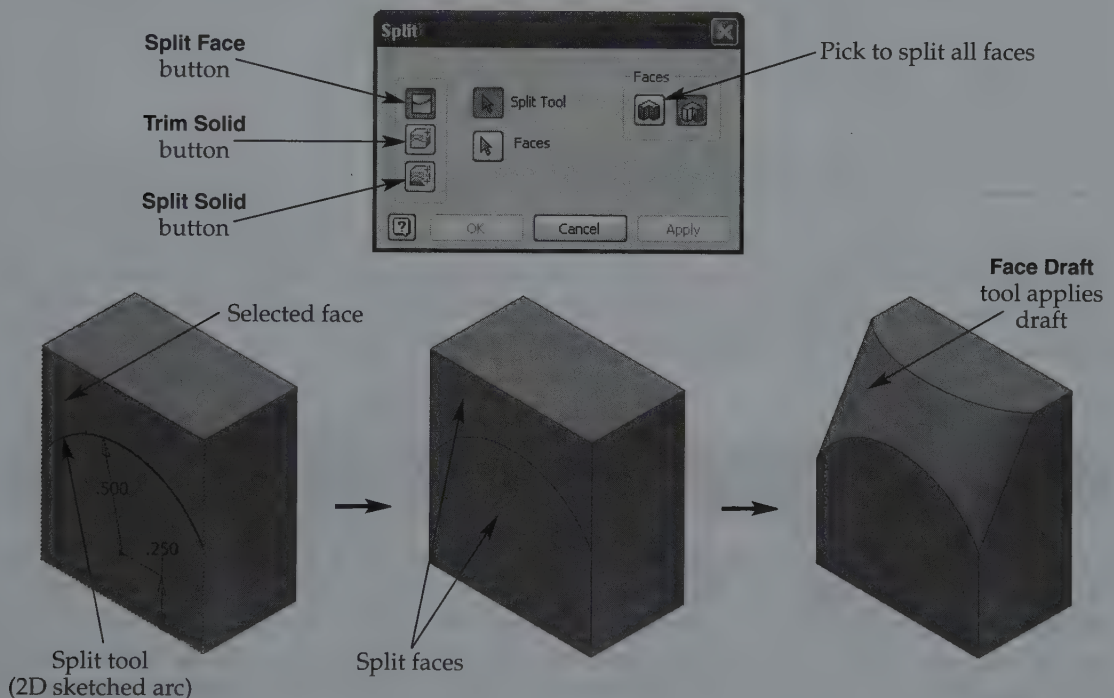
Split faces

NOTE

You can access split faces using tools such as **Face Draft**, **Move Face**, and **Thicken/Offset**. In order to use the split faces for purposes such as developing sketch geometry, you must open a new sketch on the face and project the split edges.

Exercise 13-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 13-1.

Trimming a Solid

The **Split** tool also allows you to remove a portion of a solid using the **Trim Solid** button. See **Figure 13-3**. Use the **Split Tool** button to select the split tool. Then specify to the portion of the solid to be removed by picking one of the **Side to Remove** buttons. Pick the **Apply** button to create the split. Continue splitting, or pick the **OK** button, press [Esc], right-click and pick **Done**, or access another tool to exit.

You can use the **Trim Solid** function as a general modeling tool or to develop two separate parts—for example, a top half and a bottom half. The following steps describe an example of the process for creating a top half part named Top.ipt and a bottom half part named Bottom.ipt using the **Trim Solid** function.

1. Save the part with the split tool, but before applying the split. Name the file Top.ipt.
2. Use the **Split** tool to remove the bottom side of the part.
3. Use the **Save As...** tool to save the current part, Top.ipt, with the bottom section removed, using the name Bottom.ipt.
4. The second file, Bottom.ipt, should be open. Right-click on the split in the browser and select **Edit Feature**. Select the opposite **Side to Remove** button to flip the trim, and pick the **OK** button.
5. Save and close the active file.

> **NOTE**
>
> The steps previously described for developing two separate halves of a part represent one possible technique. You may develop your own techniques, depending on the application. For example, you may want to save an additional Master.ipt file representing the part without the split.

Exercise 13-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 13-2.

Figure 13-3.
Using the **Trim Solid** method in the **Split** dialog box to remove material from a solid.

Trim Solid
button

Split tool
(origin plane)
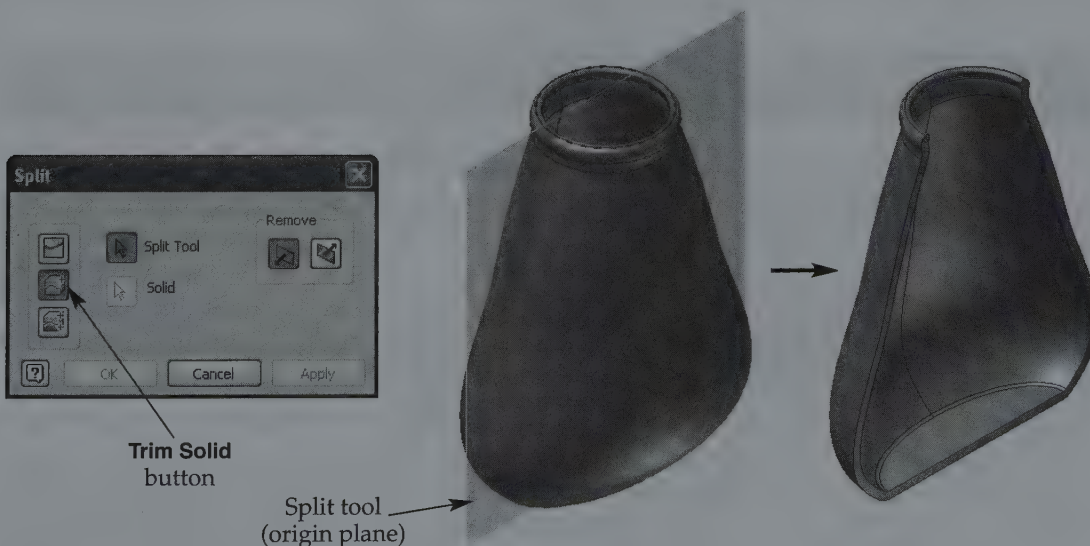
Figure 13-4.
Using the **Split Solid** method in the **Split** dialog box to create two separate solid bodies from a single solid body. Each solid is assigned a unique color to distinguish the features. By default, all solids use the same color.

Separate solid

Separate solid

Split Solid button

Split tool (origin plane)

Splitting a Solid

The **Split** tool also allows you to split a solid into two solids using the **Split Solid** button. See **Figure 13-4**. Use the **Split Tool** button to select the split tool. Pick the **Apply** button to create the split. Continue splitting, or pick the **OK** button, press [Esc], right-click and pick **Done**, or access another tool to exit. The primary function of the **Split Solid** option is to derive an assembly and two separate parts from a single part file, which is an alternative to the process described using the **Trim Solid** method.

Supplemental Material

Deriving Split Solids
For an introduction to deriving split solids to create separate parts and an assembly, go to the Student Web site (www.g-wlearning. com/CAD), select this chapter, and select **Deriving Split Solids**.

Ribbon

Model > Create

Emboss

embossing: The process of raising shapes or text off the surface of an object that has volume, such as a block.

engraving: The process of cutting into, or impressing, shapes or text into the surface of an object that has volume.

Embossing and Engraving

The **Emboss** tool allows you to replicate *embossing* and *engraving*. As shown in **Figure 13-5**, the **Emboss** tool requires a sketch profile, which you can position on a face or an offset plane. The embossing or engraving projects onto the feature, even though the sketch may not actually be on the embossed or engraved face. Access the **Emboss** tool to display the **Emboss** dialog box. See **Figure 13-6**. If a sketch includes one profile, the profile is selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile(s) to emboss and/or engrave.

Figure 13-5.
Embossed and engraved features reference a sketch profile. The sketch plane can be on a face or an offset plane.

Sketch on Plane

Embossing

.063

fx:.125

Sketch on Face

Engraving
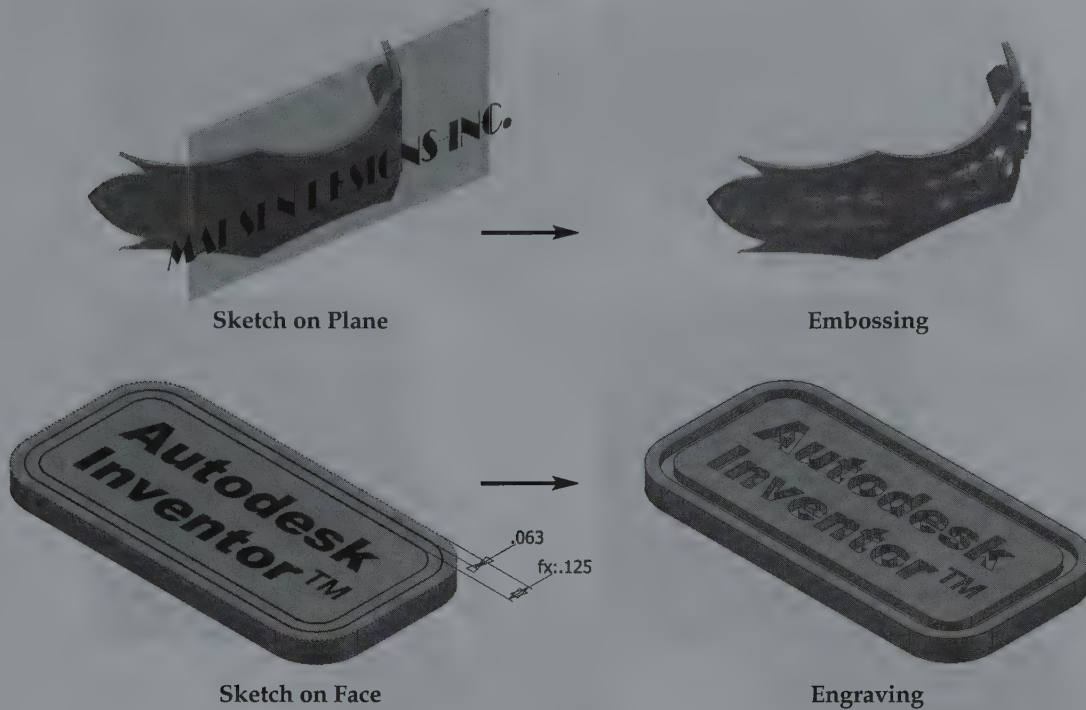
Figure 13-6.
The **Emboss** dialog box with the **Emboss from Face** option selected.

Emboss/Engrave from Plane button

Engrave from Face button

Emboss from Face button

Top Face Color button
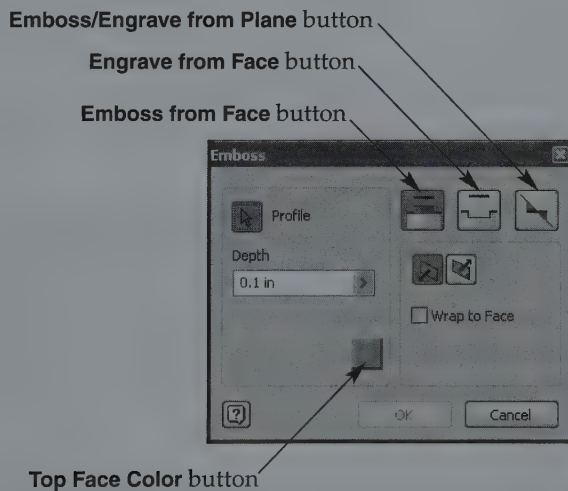
Embossing

The **Emboss from Face** button is active by default and creates an embossed feature. Specify the embossing depth using the **Depth** text box and, if needed, redefine the direction by picking the appropriate direction button. By default, if you create an embossing from a curved surface, the embossing forms perpendicular to the sketch profile. To wrap the embossing around a curved face, pick the **Wrap to Face** check box and select the face. See **Figure 13-7**. To add a unique color to the top face of the embossing, pick the **Top Face Color** button to select a color from the drop-down list in the **Color** dialog box. Pick the **OK** button in the **Emboss** dialog box to generate the feature.

Figure 13-7.
A—An example of an embossing that does not wrap to a curved face. B—Wrapping the embossing to the curved surface.

Top View   Top View

Isometric View   Isometric View

A   B
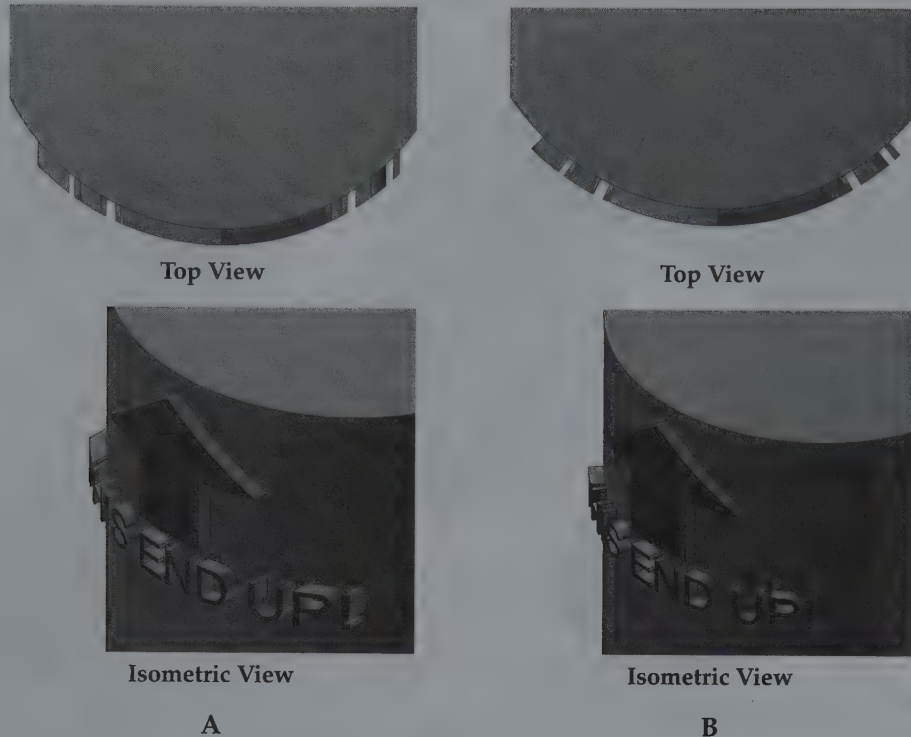
Engraving

Pick the **Engrave from Face** button to create an engraved feature. Specify the engraving depth using the **Depth** text box and, if needed, define the direction by picking the appropriate direction button. The **Wrap to Face** check box allows you to wrap the engraving around a curved face. Use the **Top Face Color** button to add a unique color to the engraved face. Pick the **OK** button to generate the feature.

Embossing and Engraving from a Plane

Pick the **Emboss/Engrave from Plane** button to emboss or engrave from the sketch plane at a specified taper angle. Specify a draft taper angle using the **Taper** text box. Then, if needed, define the direction by picking the appropriate direction button. **Figure 13-8** shows examples of using different flip options and taper angles. Use the **Top Face Color** button to add a unique color to the embossed and engraved faces. Pick the **OK** button to generate the feature.

> **NOTE**
>
> If the sketch you create extends past a curved face, you may have to pick the **Wrap to Face** check box in order to create an embossing or engraving.

Exercise 13-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 13-3.

Figure 13-8.
These parts use
the same sketch
profile, but different
emboss/engrave
settings.
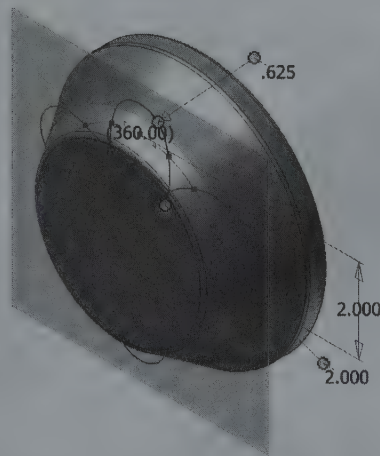
Sketch for **Emboss/Engrave from Plane** option
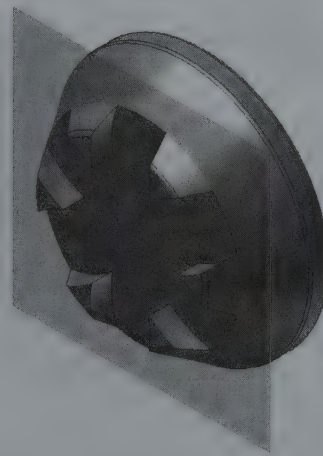
Embossing/engraving toward the model with a taper of 0°
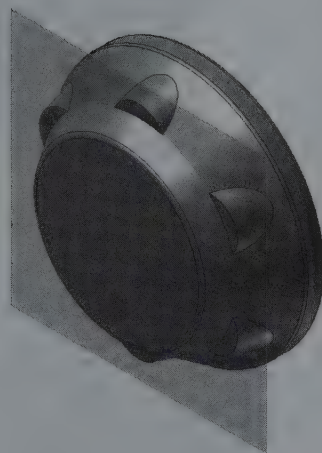
Embossing/engraving away from the model with a taper of 15°
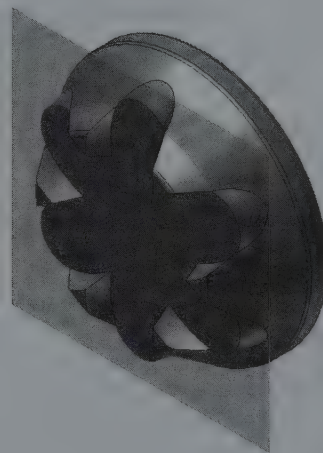
Embossing/engraving in both directions with a taper of 10°

Decals

Several applications require *decals*, including product labels, tags, art, text, or seals. See **Figure 13-9**. Decals are sketched features, created from sketches that include an image, text, or spreadsheet file. A decal can represent a surface finish, similar to applying paint, or can be a separate part with its own properties, including a part number. You can add and constrain a decal part file to multiple assemblies. For example, **Figure 13-10** shows an assembly of three individual parts. One of the components is a decal constrained, or applied, to one of the parts.

decals: Images applied to a part or assembly to display information or decorate a product.

NOTE

Adding a decal to a part, instead of creating the decal as a separate part and inserting it into an assembly, is similar to painting or changing the color of a part. Typically, you would use this technique if separate decal part properties, including an individual part number, are not required or you do not want to use the decal in different assemblies with multiple components.

Figure 13-9.
Decal examples.

Sticker

Warning Label

Decal Applied to Model
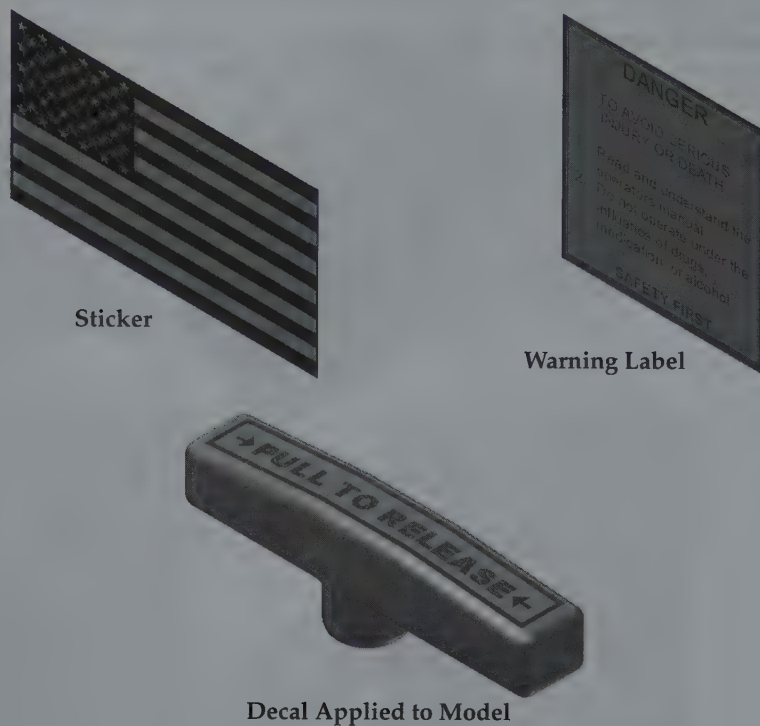
Figure 13-10.
Using a decal as a
separate part in an
assembly.

Part
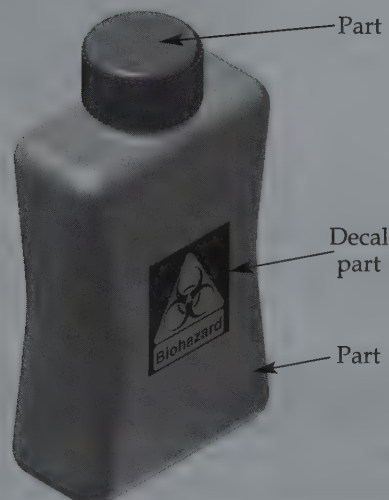
Decal
part

Part

Most existing part surfaces can receive a decal, such as the PULL TO RELEASE decal applied to the part shown in **Figure 13-9**. If you plan to create a single part decal for use in assemblies, you must produce a separate part to accept the decal. You can use a surface for this application, but a thin piece of material that represents the actual thickness of the decal, such as an extruded rectangle, may be more appropriate. See **Figure 13-11**. The part face that will receive the decal should be the same size as the decal. If the face is too small, the decal cuts off around the edges, and if the face is too large, the actual size of the sketched decal image may not display correctly.

Decal Sketching

You can position a 2D decal sketch on a face or an offset plane, as shown in **Figure 13-12**. In some cases, such as when the decal must wrap over complex geometry, it may be easier to generate a decal from a sketch offset from the feature face. A decal sketch includes an image, document, or spreadsheet file added using the **Insert Image**

Figure 13-11.
A—Extruding a rectangle to create a realistic, separate decal part using a thin solid. B—You can apply a decal to a surface, such as the surface extruded from a line shown here, but the surface contains no mass.

Extrusion thickness is the same as the label material

Surfaces have no thickness

A

B
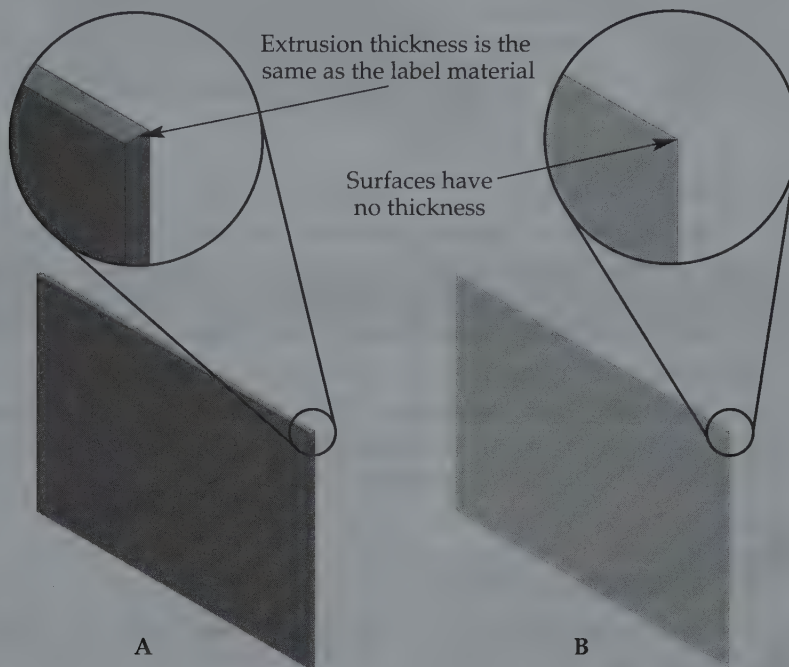
tool. When you access the **Insert Image** tool, the **Open** dialog box appears, allowing you to select the file to insert. Choose a supported image file or select a Microsoft® Word document or Microsoft® Excel spreadsheet. When inserting an image, you may find that .bmp files are supported best; other file types may not appear correctly. Pick the **OK** button to insert the image into the graphics window. Pick a location for the image, and continue placing copies as needed. To exit, press [Esc], right-click and pick **Done**, or access another tool.

Ribbon

Sketch
> Insert

Insert Image

An imported image has a rectangular boundary defined when the image is created. Drag a boundary corner to resize the image, or drag the entire image to a new location. Use constraints to specify the size and location of the image. However, notice that the image controls some dimensions. Right-click on the image and pick **Properties** to access the **Image Properties** dialog box. See **Figure 13-13**. Use the **Rotate** and **Mirror** buttons to rotate or mirror the image in reference to the specified image dimensions and constraints. Pick the **Use Mask** check box to display the image with a transparent background.

Figure 13-12.
A—An example of sketching on a thin extrusion face. B—An example of sketching on a work plane tangent to a cylinder. Notice how most of the sketch does not touch the face to receive the decal.

A

B
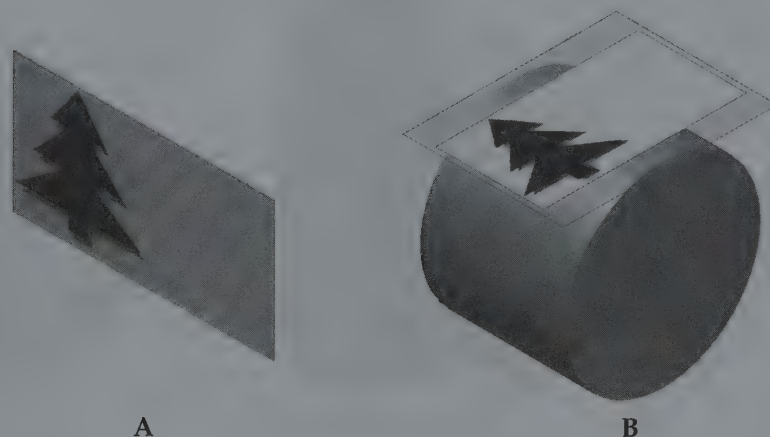
Figure 13-13.
The **Image Properties** dialog box.

Pick to rotate the image

Pick to mirror the image

Image Properties

Orientation   Transparency
                     ☐ Use Mask

OK

Using the Decal Tool

Ribbon
Model
> Create
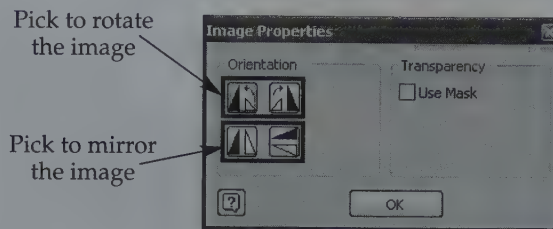
Decal

Once you finish the sketch, access the **Decal** tool to create the decal feature using the **Decal** dialog box. See **Figure 13-14**. The **Image** button is active by default, allowing you to pick the decal image sketch. The **Face** button then activates, allowing you to choose the face on which to apply the decal. When placing a decal on a curved or irregular shape, select the **Wrap to Face** check box to correctly fold, or project, the image around a nonlinear face. See **Figure 13-15**. To wrap the decal to multiple adjacent faces, such as over an edge or on a chamfer, select the **Chain Faces** check box. See **Figure 13-16**. Pick the **OK** button to generate the decal.

Exercise 13-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 13-4.

Figure 13-14.
The **Decal** dialog box.

Pick to select the decal image sketch

Pick to select the face on which the decal will be applied

Decal

⬚ Image   ☐ Wrap to Face

⬚ Face   ☑ Chain Faces

OK   Cancel
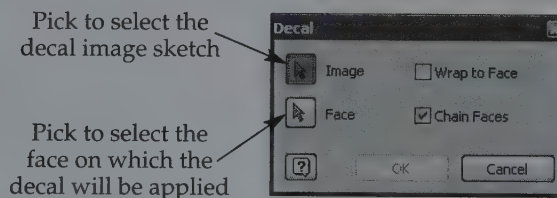
Figure 13-15.
Compare the distortion of a decal placed with and without wrapping to a curved face.

Wrap to Face is off   **Wrap to Face** is on

Figure 13-16.
Placing a decal with and without chaining faces.

Chain Faces is off   **Chain Faces** is on
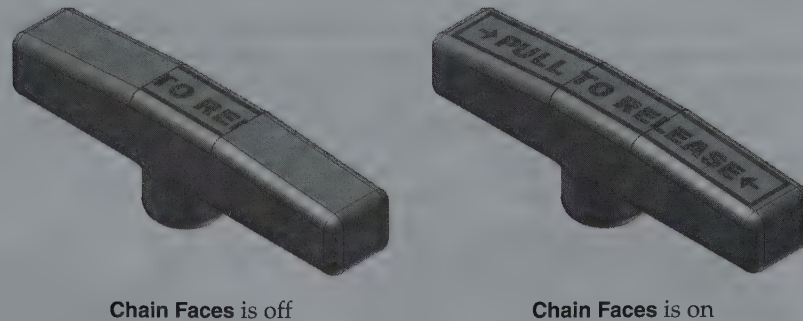
Surface Tools and Techniques

You have already discovered how to create surfaces using a variety of tools, including the **Extrude**, **Revolve**, **Coil**, **Thicken/Offset**, **Loft**, and **Sweep** tools. To create a surface using these features, pick the **Surface** output type. Like work planes, surfaces help you develop model geometry that is difficult or impossible to produce otherwise. **Figure 13-17** shows a common surface application—creating a termination area for other features. **Figure 13-18** shows a similar application. This example shows splits and thickenings that reference surfaces to further develop the model.

You can create surfaces using closed loop sketch geometry, such as the circle shown in **Figure 13-18**, or open loop sketches, such as the line and arc shown in **Figure 13-17**. Modify surface sketches as you would any other sketch. You can also add placed features, such as fillets and chamfers, to surface edges, as shown in **Figure 13-19**, and pattern surfaces using feature pattern tools.

Surfaces appear semi-translucent by default. To display surfaces with an opaque appearance, much like other model geometry, right-click on the surface in the browser and deselect **Translucent**. To hide a surface, right-click on the surface in the browser and deselect **Visibility**.

> **NOTE**
>
> The **Construction** area in the **Part** tab of the **Application Options** dialog box includes an **Opaque Surfaces** check box that controls the default translucent or opaque display of surfaces.

Figure 13-17.
An example of using a surface to control the shape of an extrusion.

Sketch placed on work plane surface

Surface extruded from a sketched arc

Cut extrusion to surface
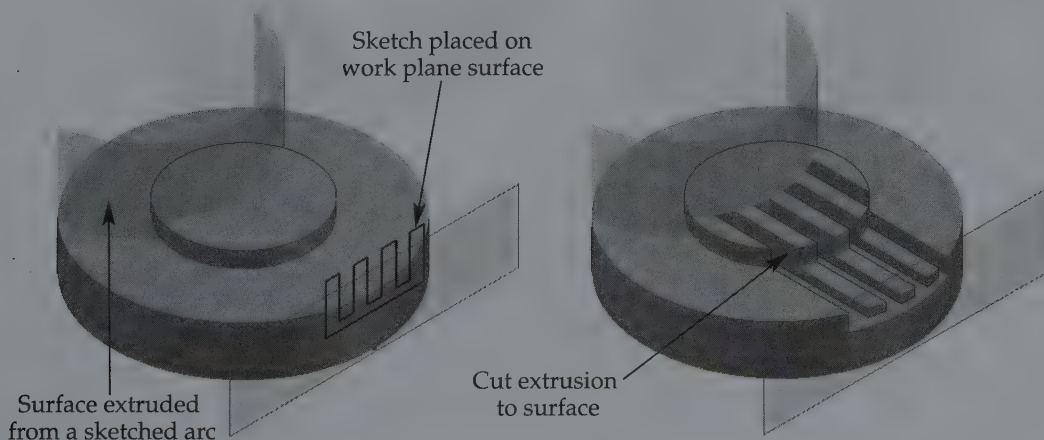
Figure 13-18.
A—Extruded and revolved surfaces created for construction purposes. The sketches are visible for reference. B—A circular patterned extrusion that extends past the surfaces. C—Using two separate split tools and the **Split Part** method to remove unnecessary geometry. D—Two thickening operations complete the part. Surface visibility is off.

Revolved surface
Extruded surface
Surface extrusion sketch
Circular pattern

1.000
.625
.125
.080
.200

Surface revolution sketch

A

Extrusion

B

First split tool
Second split tool

C

Thickened from surface

D
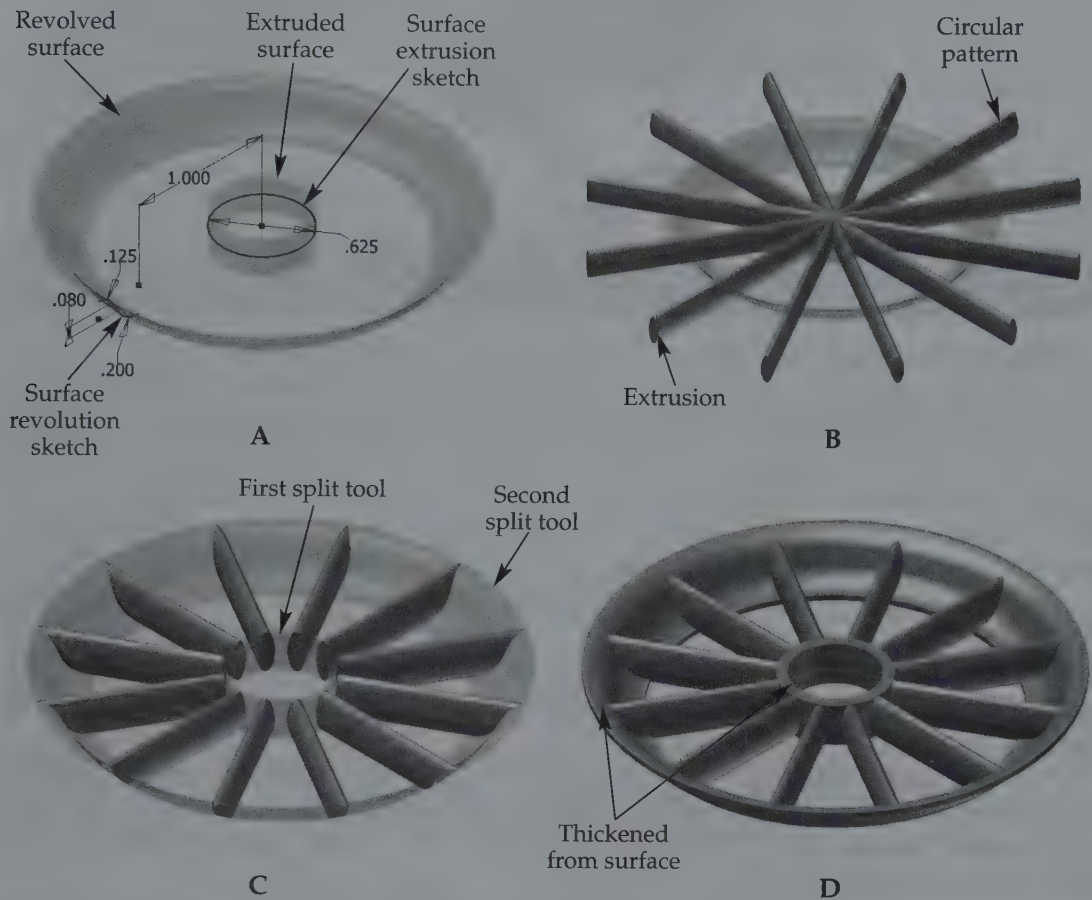
Figure 13-19.
Adding placed features, radii, and chamfers to a surface.

Original Surface

Surface with Placed Features
(highlighted for clarity)
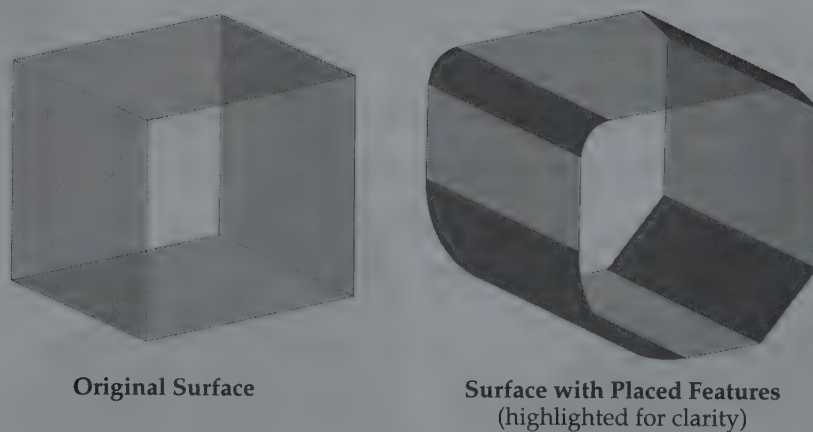
Additional Surface Tools

For information about using other common surface tools and options, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **Additional Surface Tools**.
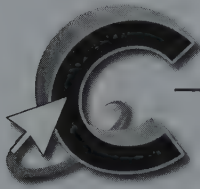
Exercise 13-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 13-5.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. Define *split* and give an example of its use.
2. Regardless of the technique you use to split a feature, what must you do first?
3. Briefly describe embossing.
4. How is engraving different from embossing?
5. Describe decals and give an example of their use.
6. Briefly explain the benefits of creating a decal as a separate part file.
7. What is a decal sketch?
8. Briefly explain how you can adjust the size and properties of an imported image.
9. What should you do if an image does not project correctly when you are placing a decal on a curved or irregular shape?
10. What is the main purpose of surfaces in Inventor?

Problems

1–2 Instructions:

- *Open the specified part file, and save the file using the given name.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **File:** EX13-1.ipt
   Save as: P13-1.ipt
   Title: TITLE PLATE
   Specific Instructions: Open a sketch on the face shown and sketch the geometry text shown. Emboss the sketched text .0625″, and add the Blue top face color. Open a sketch on the same face and create the text shown without using a text box. Engrave the sketched text .03″, and add the Red top face color. The final part should look like the part shown.

Selected

Selected

Inventor and Its Applications

2010
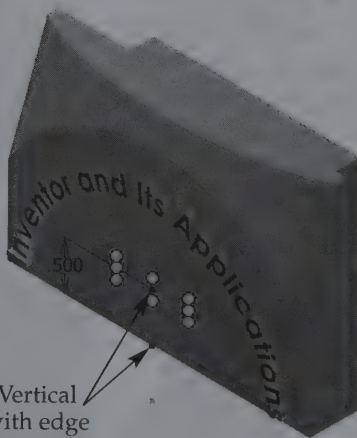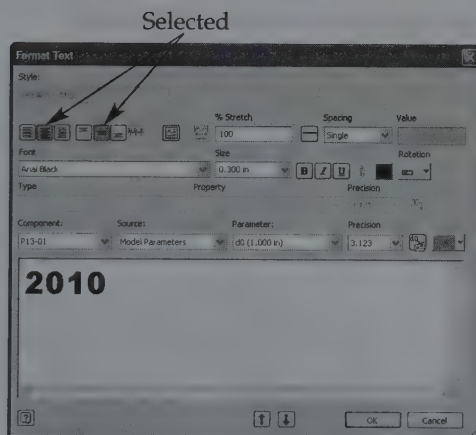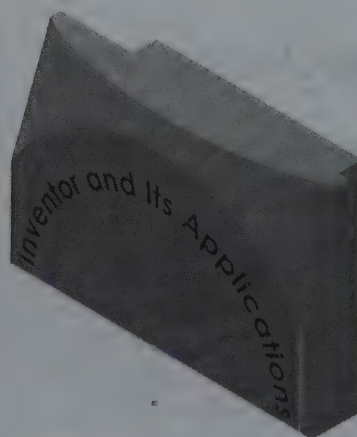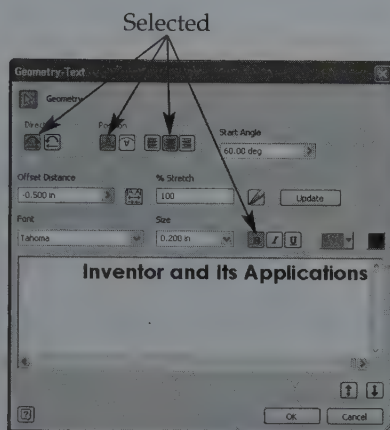
Vertical
with edge
midpoint

2. **File:** P11-7.ipt
   Save as: P13-2.ipt
   Title: MUG
   Specific Instructions:
   A. Go to the Student Web site (www.g-wlearning.com/CAD) and select **Chapter 13** in the **Chapter Materials** drop-down list. Download the GW_logo.bmp image file. Save the file in a location where you will be able to find it later.
   B. Create a work plane tangent to the cylindrical face shown by picking the face and the XY plane. Open a sketch on the work plane and insert the GW_logo. Constrain the image as shown. Create a decal of the image, wrapped to the mug face.
   C. Create a work plane tangent to the cylindrical face shown by picking the face and the XY plane. Right-click on the work plane and select **Flip Normal** to flip the work plane. Open a sketch on the work plane and insert the GW_logo. Constrain the image as shown. Create a decal of the image, wrapped to the mug face.
   D. Turn off the visibility of the work planes. The final part should look like the part shown.

3–8 Instructions:

- Create sketches of the following objects.
- Develop sketch geometry from the projected center point.
- Infer as many geometric constraints as possible and appropriate.
- Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.
- Use the information in the status bar to create objects the approximate size given by the dimensional constraints.
- Add the dimensional constraints shown.
- Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.
- Follow the specific instructions for each problem to create the features.

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

3. **Title:** SOAP PATTERN TOP
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-029-01
   Project: SOAP PATTERN
   Material: Default
   Color: Default
   Save as: P13-3.ipt

   Specific Instructions: Sketch a 4″× 6″ rectangle on the XZ plane. Extrude the sketch 3″ in the positive direction. Offset a work plane −1.5″ from the top face of the extrusion. Use the **Split** tool to split the extrusion using the **Split Part** method and using Work Plane1 as the split tool. Remove the bottom section. Open a new sketch on the bottom face of the split feature. Offset the projected edges of the rectangle .5″ as shown. Cut-extrude the sketch 1″ in the positive direction. Add fillets using a radius of .5″. The final part should look like the part shown.

.500
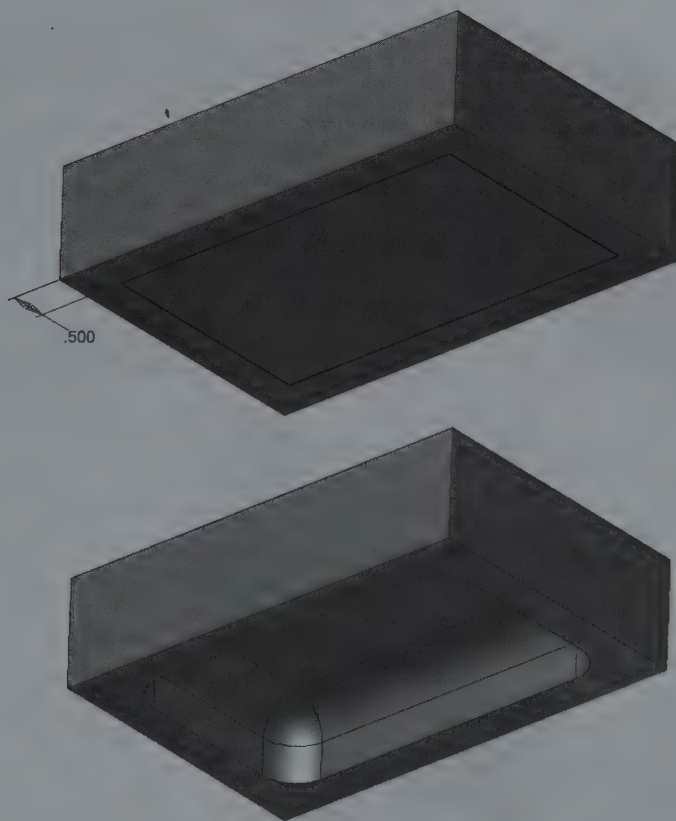
4. **Title:** SOAP PATTERN BOTTOM
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-029-02
   Project: SOAP PATTERN
   Material: Default
   Color: Default
   Save as: P13-4.ipt
   Specific Instructions: Open P13-3.ipt and save it as P13-4.ipt. In the P13-4.ipt file, delete Extrusion2 and Fillet1. Right-click on Split1 and select **Edit Feature**. Pick the opposite **Side to Remove** button, and remove the top half of the extrusion. Open a new sketch on the top face of the split feature. Offset the projected edges of the rectangle .5" as shown. Cut-extrude the sketch 1" in the negative direction. Add fillets using a radius of .75". The final part should look like the part shown.
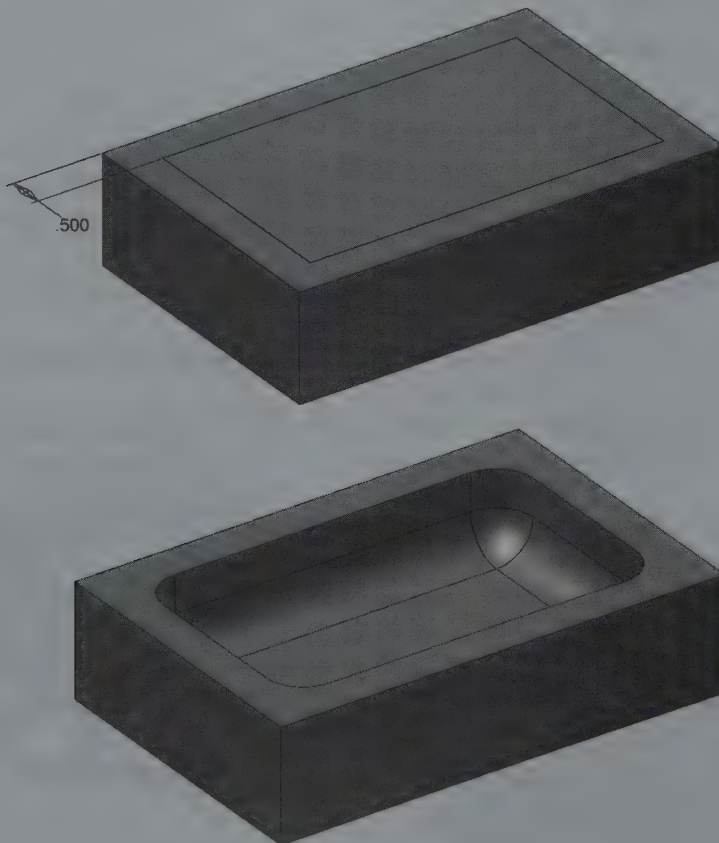
5. **Title:** DECAL IMAGE
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-030-01
   Project: DECAL IMAGE
   Material: Default
   Color: Default
   Save as: P13-5.ipt
   Specific Instructions: Access an image file, such as a .bmp file, and create a separate part decal using the information described in this chapter.

6. **Title:** DECAL DOC
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Number: IAA-031-01
   Project: DECAL DOC
   Material: Default
   Color: Default
   Save as: P13-6.ipt
   Specific Instructions: Access a .doc or .xls file and create a separate part decal using the information described in this chapter.

7. **Title:** FLOW INTERRUPTER
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-032-01
   Project: HIGH FLOW VALVE
   Material: Stainless Steel
   Color: As Material
   Save as: P13-7.ipt
   Specific Instructions:
   A. Open a 2D sketch on the XZ plane and sketch a .625″ circle with a center point coincident to the projected center point. Surface extrude the sketch .25″ midplane.
   B. Open a sketch on the YZ plane and sketch the geometry shown. Surface-revolve the sketch fully around the Y axis.
   C. Open a sketch on the XY plane and sketch the ellipse shown. Solid-extrude the sketch 2″ in the positive direction.
   D. Use the **Circular Pattern** tool to pattern the extruded ellipse. Select the Y axis as the rotation axis, specify a count of 12, a fitted position method, and a 360° angle.
   E. Use two separate **Split** tool operations to trim the extrusions as shown, using the surfaces as split tools.
   F. Use the **Thicken/Offset** tool to thicken the extruded surface shown .0625″ toward the center of the model. Select the inner surface created by the extrusion.
   G. Use the **Thicken/Offset** tool to thicken the revolved surfaces shown .03125″ away from the center of the model. Select the three outer surfaces created by the revolution.
   H. Turn off the visibility of the extruded and revolved surfaces. The final part should look like the part shown.

Coincident

.125

Vertical

.080

Line tangent
to arcs   .200

Vertical

Coincident

1.000

Projected
center point

Projected surface
extrusion edges

Projected center point

.125

−.050
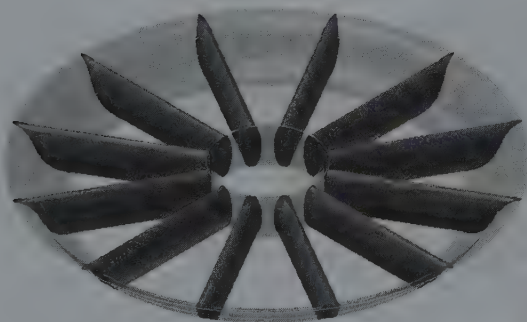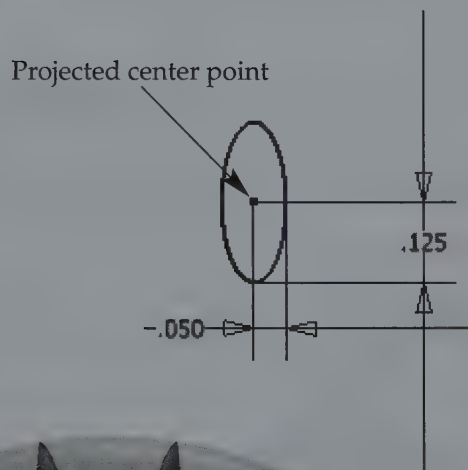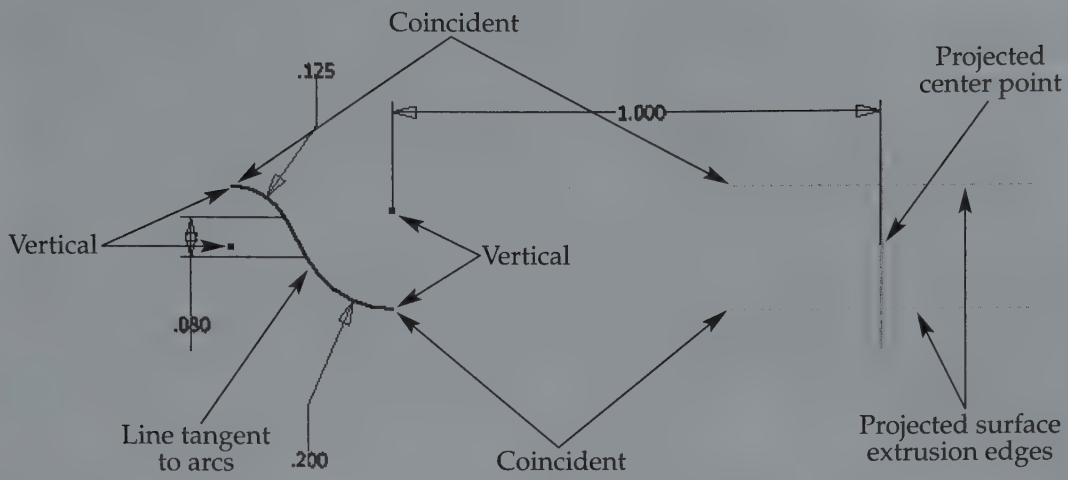
8. **Titles:** CALCULATOR FACE and CALCULATOR BASE
   Units: Inch or Metric
   Template: Part-IN.ipt or Part-mm.ipt
   Part Numbers: IAA-033-01 (face) and IAA-033-02 (base)
   Project: CALCULATOR SHELL
   Material: ABS Plastic
   Color: Gray
   Save as: P13-8FACE.ipt (face) and P13-8BASE.ipt (base)
   Specific Instructions: Create a calculator face and base similar to the face and base shown. Generate the parts using the techniques described in this chapter and previous chapters. Although you may develop your own methods for creating the calculator parts, you may want to refer to the following information for assistance.
   A. Extrude the entire calculator (face and base intact).
   B. Add fillets.
   C. Add work planes or a sketch to define the split parting line.
   D. Save the part as P13-8FACE.ipt.
   E. Split the feature and remove the bottom (base) section.
   F. Create button slots by cut-extruding rectangles. Use a rectangular pattern to create certain button slots.
   G. Place holes and add fillets.
   H. Save the part as P13-8BASE.ipt and edit the split to remove the top (face) section.
   I. Extrude the base feet.

14

iFeatures

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create and insert iFeatures.
✓ View and manage iFeatures.

An *iFeature* is an example of a *catalog feature* that you save for repeated use. **Figure 14-1** shows an example of an iFeature, consisting of several features, added to an extrusion. You have the option of inserting an iFeature using the original size and position of features, or adjusting the features according to unique design requirements.

iFeature: A stored feature or group of features you can insert in a part as a feature.

catalog feature: A feature, part, or assembly stored in a catalog for insertion into a part model as a feature.

Creating an iFeature

An iFeature is a separate .ide file that you create by extracting existing features. You can then reference the iFeature as a design element in other parts. To create an iFeature, first access the part that contains the features you want to extract. You can extract sketched features and *included* placed features. For example, to extract the round and fillet shown in **Figure 14-1**, you must include the extruded cylinder.

Figure 14-1.
A—A model ready to receive an iFeature. B—Placing a group of standard features as a single iFeature. This iFeature includes an extrusion, threaded hole, chamfer, fillet, and round. C—The final part built from an extrusion and iFeature.

A   B   C
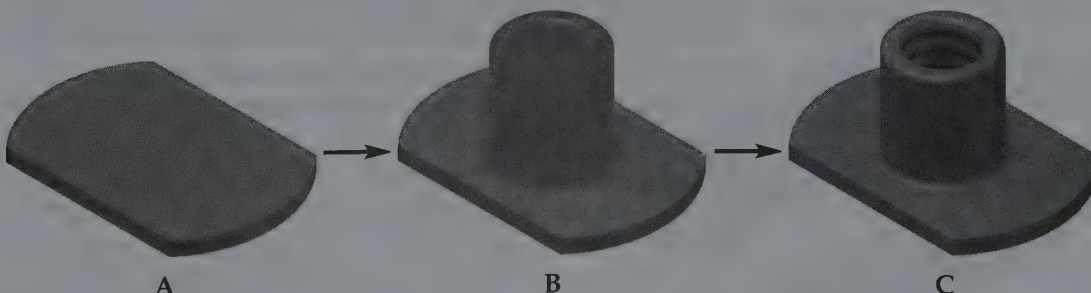
In order to extract a hole feature without including the features in which the hole cuts, you must use the **From Sketch** placement option to create the hole.

Use the **Parameters** dialog box to modify the names of parameters you plan to have the ability to adjust. Each name should reflect the dimension it controls. For example, change d1 to Cylinder_Diameter and d2 to Cylinder_Height.
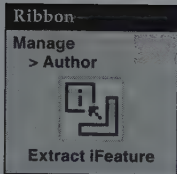
Ribbon

Manage
> Author

Extract iFeature

Access the **Extract iFeature** tool to extract an iFeature using the **Extract iFeature** dialog box. See **Figure 14-2**. Pick the **Standard iFeature** radio button to extract a standard iFeature, or pick the **Sheet Metal Punch iFeature** radio button to create a sheet metal punch iFeature and enable options in the **Manufacturing** and **Depth** areas. You will explore sheet metal punch iFeatures later in this textbook.

Select the features to extract from the graphics window or the browser. Inventor automatically includes geometrically dependent features, such as a chamfer on a hole edge. Selected features and associated size and position parameters are displayed in

Figure 14-2.
Extracting a cut extrusion using the **Extract iFeature** dialog box. In this example, a cut extrusion is set to extract. Refer to this figure as you learn to extract iFeatures.

Pick to add the selected size parameter or position geometry

Selected feature to extract

Pick to remove the selected size parameter or position geometry
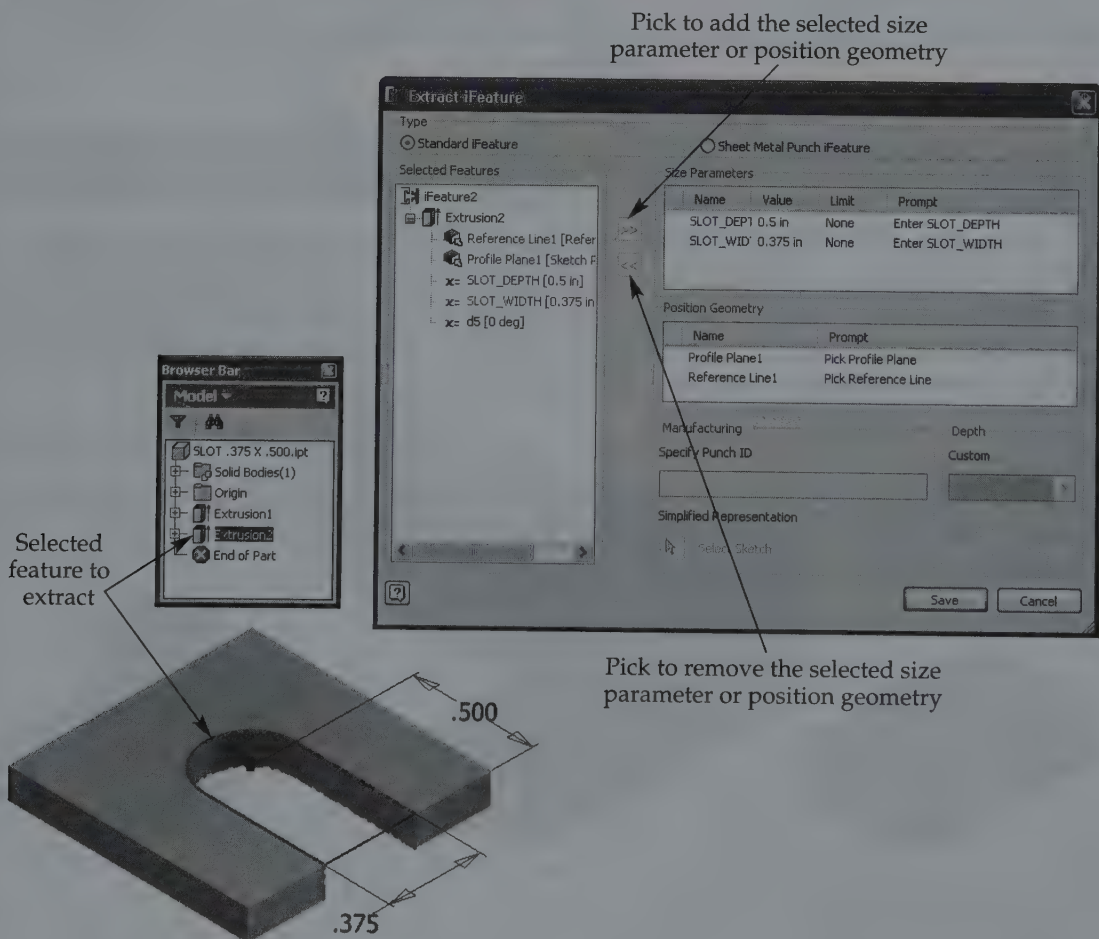
the **Selected Features** list box. To remove a feature from the extraction, right-click on the feature in the list and select **Remove Feature**. Next, you must set size and position characteristics to ensure that you can adjust the iFeature size, if necessary, and correctly position the iFeature during insertion.

Size Parameters

You can modify feature parameters when you insert an iFeature if you add the parameters to the **Size Parameters** area. Parameters that you previously renamed using the **Parameters** dialog box are automatically added to the **Size Parameters** list. To add default named parameters, pick the parameter and select the **Add Parameter** button, double-click the parameter inside the **Selected Features** area, or right-click on the parameter and select **Add Parameter**. A **Remove Parameter** option is also available.

Adjust the parameter name, value, limit, and prompt as needed using options in the corresponding columns. The **Name** column provides an opportunity to name the parameter according to the dimension it controls, if you did not already do so in the **Parameters** dialog box. The **Value** column identifies the original dimension of the parameter. When you insert an iFeature, a prompt instructs you on the parameter value operation, such as **Enter distance.** Use the **Prompt** column to modify the default prompt to a more understandable prompt. For example, change **Enter distance** to **Enter hole depth**. When you finish adjusting a name, value, or prompt, pick the pencil icon or anywhere outside of the **Size Parameters** area.

Size Limits

The **Limit** column allows you to place limits on the size changes that can occur when you insert an iFeature. The default setting is **None**, which places no limit on the size you can specify, other than those controlled by design constraints. Select an option from the drop-down list corresponding to the parameter you want to limit. When you specify a size limit, Inventor only allows you to select a value within limits during iFeature insertion.

Choose the **Range** option to assign a limited size range using the **Specify Range** dialog box. See **Figure 14-3A**. Modify the default value using the **Default** text box, which is the same as changing the value in the **Value** column. Use the **Minimum** text box and drop-down list to define a value less than, less than or equal to, or infinitely less than the default value. Use the **Maximum** text box and drop-down list to define a value greater than, greater than or equal to, or infinitely greater than the default value.

Select the **List** option to create a list of possible sizes using the **List Values** dialog box. See **Figure 14-3B**. The current default value appears in the list box. To add an optional value, select the **Click here to add value** button and enter the value. Add as

Figure 14-3.
A—Select the **Range** option to access the **Specify Range** dialog box. B—Pick the **List** option to access the **List Values** dialog box.

Less-than range
drop-down list

Greater-than range
drop-down list

A

Default value
drop-down list

B
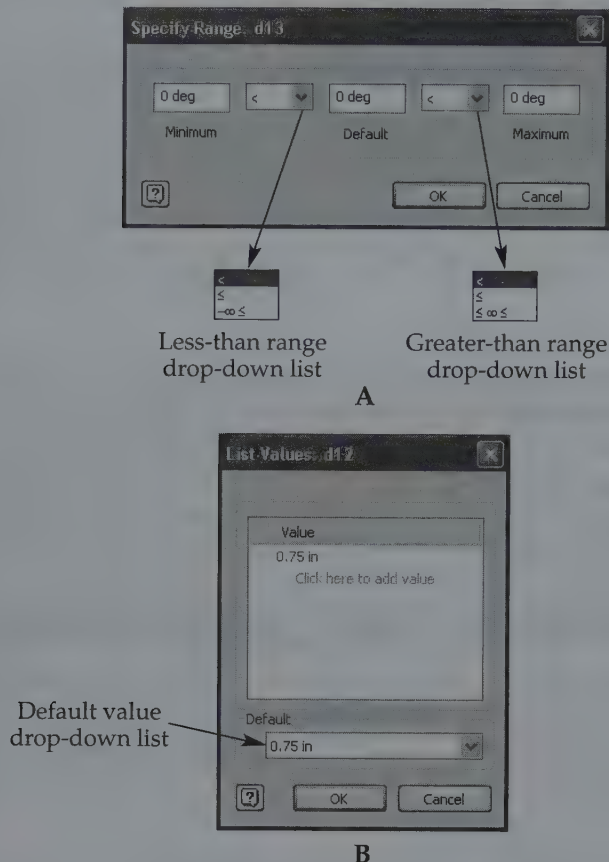
many optional values as permissible for assignment to the parameter. Assign one of the new values as default using the **Default** drop-down list, which is the same as changing the value in the **Value** column.

Position Geometry

Use the **Position Geometry** area to specify how iFeature geometry is positioned during insertion. During insertion, you will be able to position the iFeature using only reference items, such as points, edges, axes, and planes that appear in **Position Geometry** area. For example, Profile Plane1 is the reference to a sketch profile. When you insert the iFeature, you must select a face or plane to link with Profile Plane1 in order to locate the iFeature.

All reference geometry associated with selected features appears in the **Selected Features** area with the matching icon. For example, Extrusion2 in **Figure 14-2** includes Reference Line1 that forms when coincident constraints connect the square corners of the slot with the projected edge. Extrusion2 also includes Profile Plane1 that identifies the sketch plane. This position geometry is required to locate the iFeature as in the original model. To add position geometry, pick geometry in the **Selected Features** area and select the **Add Geometry** button, double-click on the geometry, or right-click on the geometry and select **Expose Geometry**. A **Remove Geometry** option is also available.

Adjust the position geometry name and prompt as appropriate. If two features reference the same position geometry, such as two extrusions sharing the same sketch plane, the position geometry is combined. If you want the ability to adjust the location of each sketch plane when you insert the iFeature, right-click on combined position

geometry and select **Make Independent**. To combine the same position geometry, right-click on one of the geometry references, select **Combine Geometry**, and then pick the other geometry reference.

Finalizing the iFeature

After you specify all options in the **Extract iFeature** dialog box, pick the **Save** button to access the **Save As** dialog box. Save the iFeature in an appropriate location on your computer or the network. The default save location corresponds to the **iFeature User Root** specified in the **iFeature** tab of the **Application Options** dialog box. You will learn about this location and other iFeature settings later in this textbook.

Exercise 14-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 14-1.

Exercise 14-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 14-2.

Exercise 14-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 14-3.

Inserting iFeatures

Access the **Insert iFeature** tool to insert an iFeature using the **Insert iFeature** dialog box. See **Figure 14-4**. Pick the **Browse...** button to use the **Open** dialog box to locate an iFeature anywhere on your computer or the network. The **Position** area is active and a preview of the iFeature appears in the graphics window. See **Figure 14-5**. The prompt guides you through the process of positioning the iFeature. Pick appropriate

Ribbon

Manage
> Insert

Insert iFeature

Figure 14-4.
The **Insert iFeature** dialog box.

Insert iFeature tree structure

Pick to select an iFeature to insert
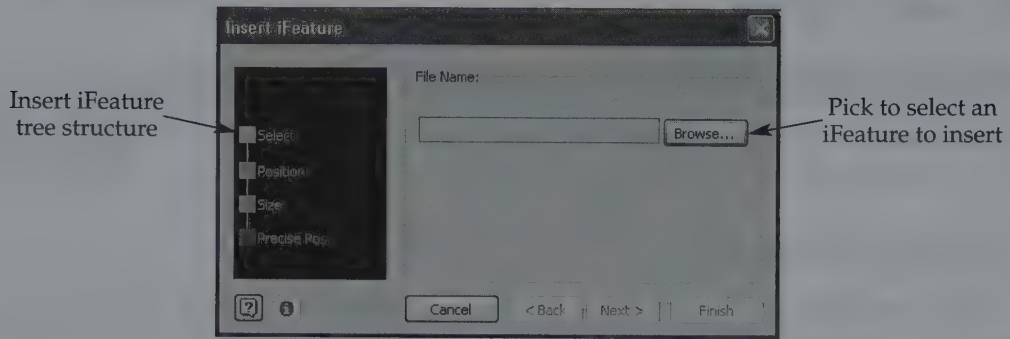
Figure 14-5.
The **Insert iFeature** dialog box with the **Position** area active. The **Rotate** and **Move** icons are visible on the inserted iFeature.

Pick to flip direction

Specify a rotational angle

Selected reference plane

Selected reference edge

Pick on the **Rotate** icon to rotate

Pick on the **Move** icon to relocate
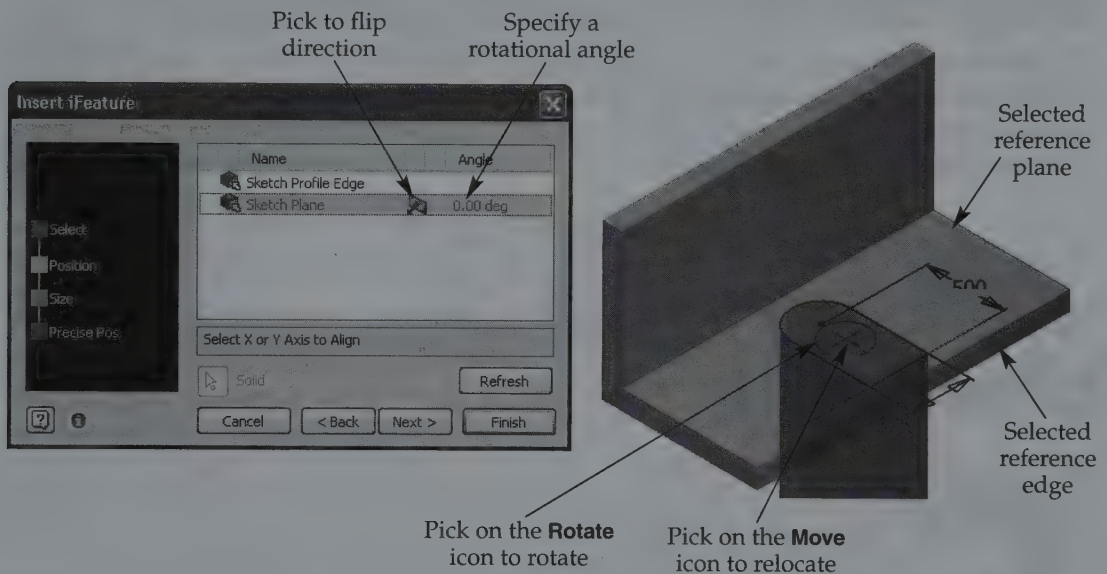
geometry in the host model, according to the prompt and the position geometry, to link the iFeature to the model. When you position a plane, you can use the **Flip** button to flip the side, the **Move** icon to move the iFeature, and the **Rotate** icon or text box to rotate the iFeature.

Pick the **Next** button or select **Size** on the tree structure to activate the **Size** area. See **Figure 14-6**. Size parameters specified during the iFeature definition appear in the **Size** list box. The prompts guide you through the iFeature sizing process. If you specified the **None** size limit option, enter any appropriate value. If you specified the **Range** size limit option, enter a value within the range. If the value is not within the limits, the value is red. If you specified the **List** size limit option, choose a value from the drop-down list.

> **NOTE**
>
> Pick the **Refresh** button in the **Insert iFeature** dialog box whenever you reposition or resize the iFeature to preview the changes.

Figure 14-6.
The **Size** area of the **Insert iFeature** dialog box.
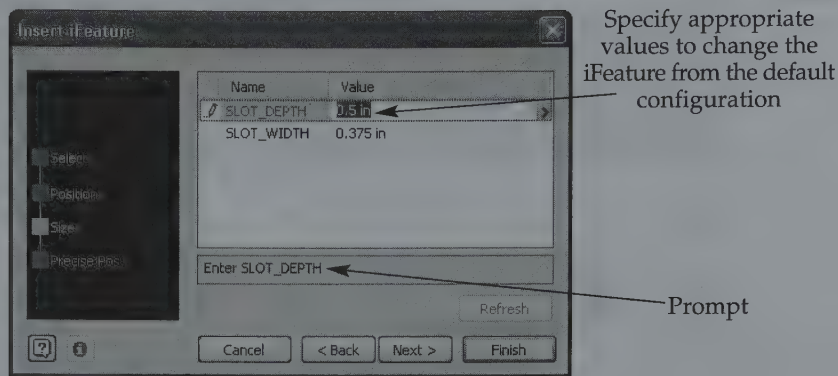
Specify appropriate values to change the iFeature from the default configuration

Prompt

Pick the **Next** button or select the **Precise Pos.** option on the tree structure to activate the **Precise Position** area. Pick the **Activate Sketch Edit Immediately** radio button to open the iFeature sketch after you complete the **Insert iFeature** dialog box, allowing you to position the iFeature precisely using sketch constraints. The iFeature sketch does not automatically open if you select the **Do Not Activate Sketch Edit** radio button. However, you can edit the sketch later to constrain sketch geometry to the host model.

Once you select, position, and size the iFeature, pick the **Finish** button to place the iFeature and, if specified, activate the sketch. However, you are not required to size the iFeature. You can pick the **Finish** button after you select and position the iFeature to bypass the other insertion options.

> **NOTE**
>
> Pick the **Back** button in the **Insert iFeature** dialog box to modify an insertion option. You can also choose any of the options on the tree structure to return to and modify an insertion option.

Exercise 14-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 14-4.

Placing Catalog iFeatures

The default location for iFeatures is the Catalog folder, found in the path: C:\Program Files\Autodesk\Inventor 2010\Catalog, depending on system settings. Inventor navigates to this folder by default when you extract or insert an iFeature. Although you can extract or insert an iFeature at any location, for the most efficient use of iFeatures, setup a catalog system similar to the default Catalog folder.

The flyout in the **Insert** panel of the **Manage** ribbon tab, shown in **Figure 14-7**, provides a visual method to insert iFeatures located in the iFeature catalog system. Pick an iFeature from the flyout to insert. Options on the **iFeature** tab of the **Application Options** dialog box control where Inventor looks to find catalog iFeatures. Any iFeatures found in folders specified in the **iFeature root, iFeature user root**, and **Sheet metal punches root** areas appear in the flyout.

Figure 14-7.
Inserting an
iFeature using the
flyout in the **Insert**
panel of the **Manage**
ribbon tab.

Pick to insert
the iFeature

Pick to access
the iFeature
viewer
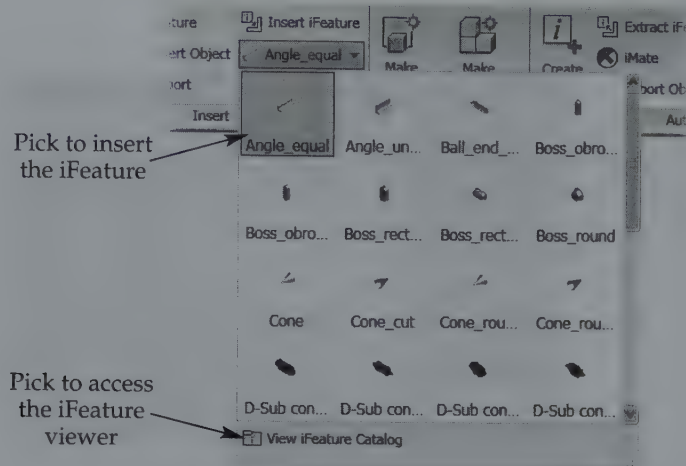
PROFESSIONAL TIP

Change the location of iFeature files using the settings in the **iFeature** tab of the **Application Options** as desired. For example, you might place the iFeatures in a network folder to allow members of the design team access to commonly used design elements.

Managing iFeatures

iFeature reference:
An iFeature inserted into a model that references an iFeature definition or .ide file.

Edit an *iFeature reference* to explore design alternatives and product changes. Whenever you insert an iFeature, a sketch extracted from an original feature forms to position the iFeature. To edit an iFeature sketch in the part environment, right-click on an iFeature or an iFeature sketch in the browser, and pick **Edit Sketch**. Edit the iFeature to redefine iFeature position geometry and size parameters specified during insertion. To edit, right-click on the iFeature or iFeature sketch in the browser and pick **Edit iFeature…**.

Editing iFeature Files

Ribbon
iFeature
> iFeature
Manage
> Insert
> Angle_Equal

View Catalog

You also have the option of editing the iFeature file (.ide) that forms when you extract an iFeature. Open an .ide file as you would any other Inventor file, using the **Open** tool or Windows Explorer. You can use the **View Catalog** tool to display and open iFeatures located in the folder specified in **iFeature viewer argument string** text box in **iFeature** tab of the **Application Options** dialog box. The viewer application specified in the **iFeature viewer** text box determines the viewer, such as Windows Explorer, that is used when you access the **View Catalog** tool.
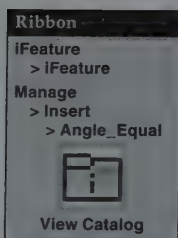
NOTE

The iFeature viewer is a quicker way to access iFeature files stored in a catalog system, which you could also find using the **Open** dialog box or Windows Explorer. You can also set up a library search path to a catalog folder in the active project.

The iFeature work environment, shown in **Figure 14-8**, includes specific .ide file tools and several of the same general tools that are available in the modeling environment,

Figure 14-8.
An example of the iFeature (.ide) file work environment.
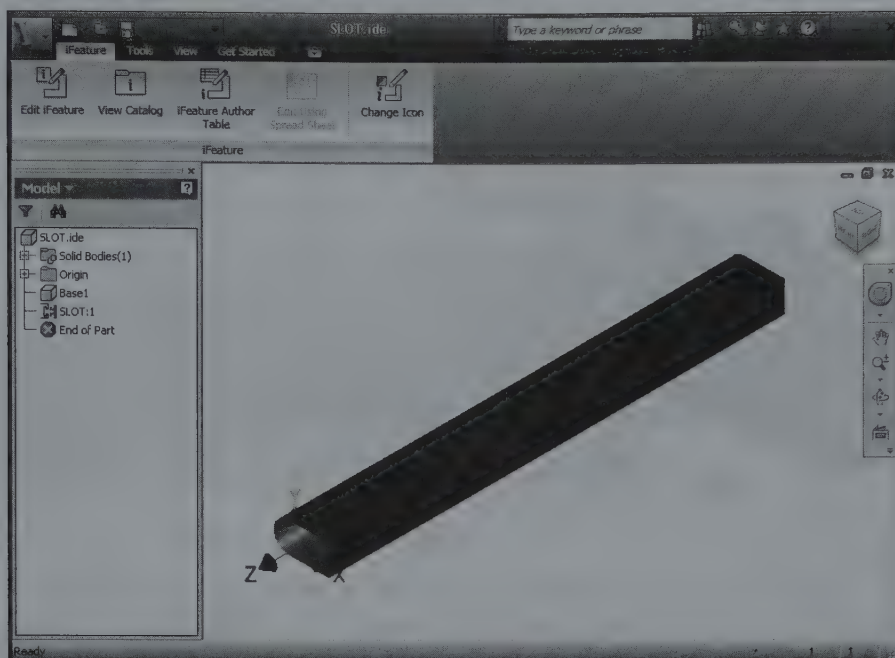
including iProperties. The **Edit iFeature** tool opens the **Edit iFeature** dialog box, which is identical to the **Extract iFeature** dialog box. Use the **Edit iFeature** dialog box in the iFeature work environment to redefine the options specified when you extracted the iFeature. Use the **View Catalog** tool to display the iFeature viewer. Use the **Change Icon** tool to adjust the appearance of the iFeature icon using the **Edit Icon** dialog box. The **iFeature Author Table** tool allows you to create a table-driven iFeature, as described later in this chapter.

Creating a Table-Driven iFeature

The standard approach to iFeature use is appropriate for many applications, allowing you to reuse feature content efficiently. An alternative is to use a *table-driven iFeature*. Each time you insert a table-driven iFeature, you can select a unique group, or row, of characteristics to create a feature reference. This approach is in some ways similar to purchasing a product from a real-world parts catalog.

Create a table-driven iFeature from an existing and open .ide file. Access the **iFeature Author Table** tool to convert the iFeature to a table-driven iFeature using the **iFeature Author** dialog box. See **Figure 14-9**. The **iFeature Author** dialog box includes a table in which you form custom records, or rows, with specific feature variables such as parameters, geometry, properties, and threads. The tabs above the table allow you to reference feature variables to include in the table as columns.

table-driven iFeature: An iFeature that allows you to create multiple variations of the original iFeature using information stored in a spreadsheet.

Parameters Tab

Figure 14-9 shows the **Parameters** tab. The list on the left side identifies all features and corresponding parameters extracted to form the iFeature. The list on the right side indicates the size parameters included during extraction or edit. Columns named after each selected parameter appear in the table. In order to add or remove a parameter column, you must edit the iFeature to add or remove parameters from the **Size Parameters** area of the **Edit iFeature** dialog box.

Figure 14-9.
The **Parameters** tab of the **iFeature Author** dialog box.

Extracted parameters

Included parameters
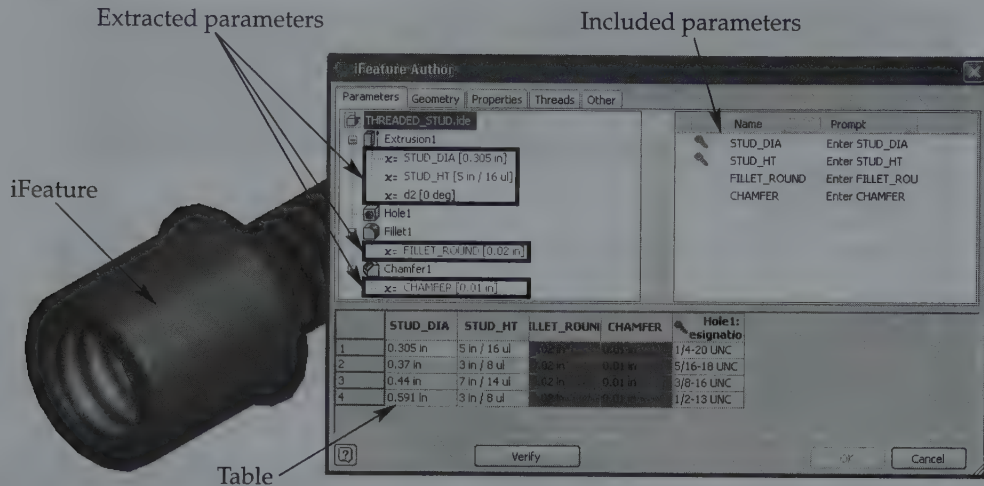
iFeature

Table

Keys

Key symbols appear next to each selected item, corresponding to a table column, in each of the **iFeature Author** dialog box tabs. Keying serves as a way to limit prompts for values that appear when you place an iFeature. For example, to have the ability to adjust the depth of a hole feature regardless of other values, add the depth parameter to the table and key the parameter. However, if the value of another column, such as an extrusion distance, always controls the hole depth, key the extrusion distance but not the hole depth. The hole depth is still a column and changes when you insert the iFeature, but a depth prompt is unnecessary.

Inventor keys all columns by default. To key specific columns, pick the key symbol or right-click on the parameter in the **Selected Items** list and select a key number from the **Key** cascading submenu. Once you assign a specific key, the default keying of all columns is lost, and you must key the other columns for which you want to receive prompts. Selected keys are blue and include a key number. Pick the number to select a different number from the list. You can key nine items in the entire table. Keys are listed in order when you insert the iFeature. Remove a key by selecting the key symbol a second time, or by right-clicking on the parameter in the **Selected Items** list and selecting **Not A Key** from the **Key** cascading submenu.

Geometry Tab

Figure 14-10 shows the **Geometry** tab. The list on the left side identifies all features and corresponding position geometry extracted to form the iFeature. The list on the right side indicates the position geometry included during extraction or edit. Use this tab to change the position geometry name and prompt. Edit the iFeature to add or remove position geometry.

Properties Tab

The **Properties** tab, shown in **Figure 14-11**, allows you to select properties identified in the **iProperties** dialog box from within the iFeature file. Use iFeature property columns whenever you need to add non-geometric information such as a description. This often helps to identify the correct iFeature record during insertion. Expand a category in the list on the left side of the tab to view related properties. Select a property and pick the **Add** button, or double-click on the item to add the property as a column in the table. **Figure 14-11** shows adding the **Description** property as a column

Figure 14-10.
The **Geometry** tab of the **iFeature Author** dialog box.

Extracted geometry   Included geometry
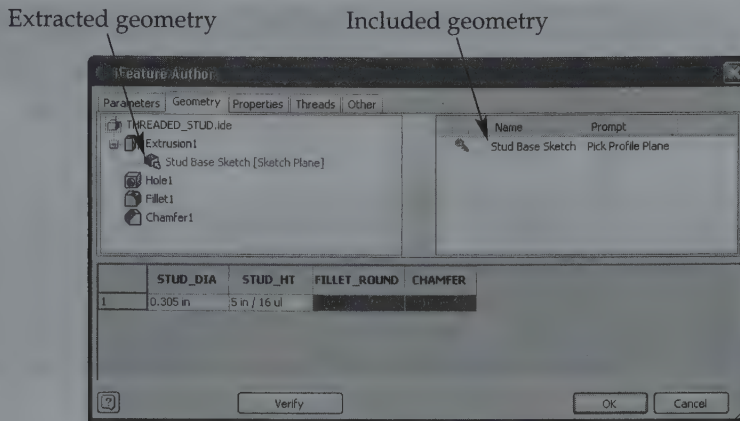
Figure 14-11.
The **Properties** tab of the **iFeature Author** dialog box.

iProperties available   **Add** button   **Remove** button   Included properties
in the iFeature file
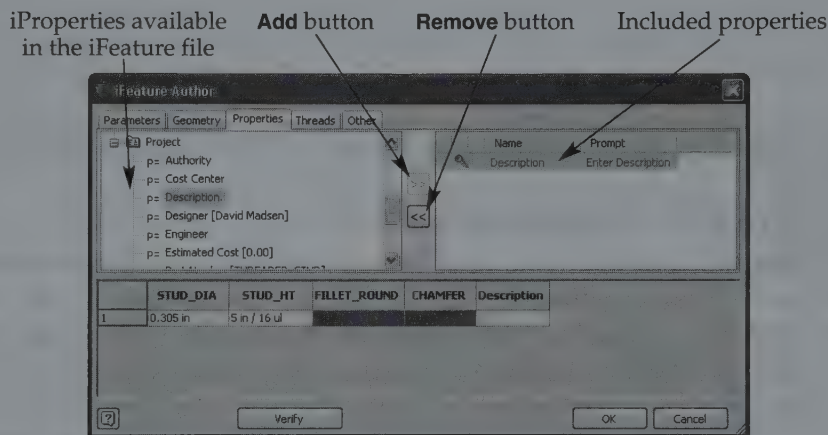
in the table. To remove a property, select the property and pick the **Remove** button, or right-click on the property and select **Delete Column**. Use keys to identify which properties prompt for a value.

Threads Tab

The **Threads** tab, shown in **Figure 14-12**, allows you to select thread specifications for iFeatures that contain threaded features. Expand a feature in the list on the left side of the tab to view thread specifications, or parameters. Select a parameter and pick the **Add** button, or double-click on the parameter to add the parameter as a column in the table. **Figure 14-12** shows adding the Designation and Class variables. All other specifications remain unchanged, regardless of the iFeature reference. The name and prompts correspond to the values defined in the threads spreadsheet file (Threads.xls). To remove a parameter, select the parameter and pick the **Remove** button or right-click on the parameter and select **Delete Column**. Use keys to identify which thread specifications are formed for a value.

> **NOTE**
>
> Adding thread specifications to a table-driven iFeature is the only way to adjust threads when using an iFeature.

Extracted thread specifications   **Add** button   **Remove** button   Included thread specifications
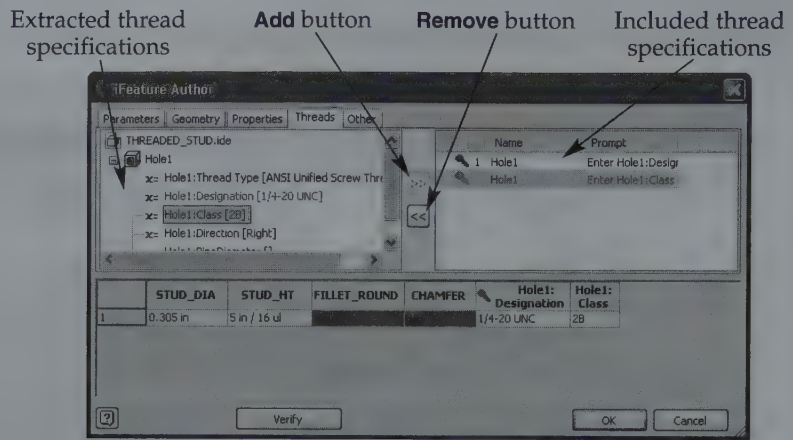
Other Tab

Use the **Other** tab, shown in **Figure 14-13**, to create custom attributes that the other tabs cannot provide. For example, define additional iFeature placement information or characteristics. To create a custom item, pick the **Click here to add value** button. Change the default NewItem0 column name and Enter NewItem0 prompt as needed. To remove a custom attribute, select the attribute and pick the **Remove** button, or right-click on the attribute and select **Delete Column**. Use keys to identify which custom attributes prompt for a value.

Adding Rows and Adjusting Columns

Columns in the iFeature table specify the feature characteristics that change between different iFeature references. Rows control the number of variations. The default row, which is green to indicate default, includes default values from each column. A single row, without assigned custom columns, means that you can only insert one variation of the iFeature. The purpose of using a table-driven iFeature is to have the ability to select from multiple types of features, which requires additional rows.

To create a row, right-click on an existing row and select **Insert Row**. A copy of the row, including cell values, appears below the selected row. Modify the values in each of the cells to reflect the desired iFeature attributes. **Figure 14-14** shows an example of

Figure 14-13.
The **Other** tab of the **iFeature Author** dialog box.

Edit icon   **Key** icon   New item name

New item prompt

New item column
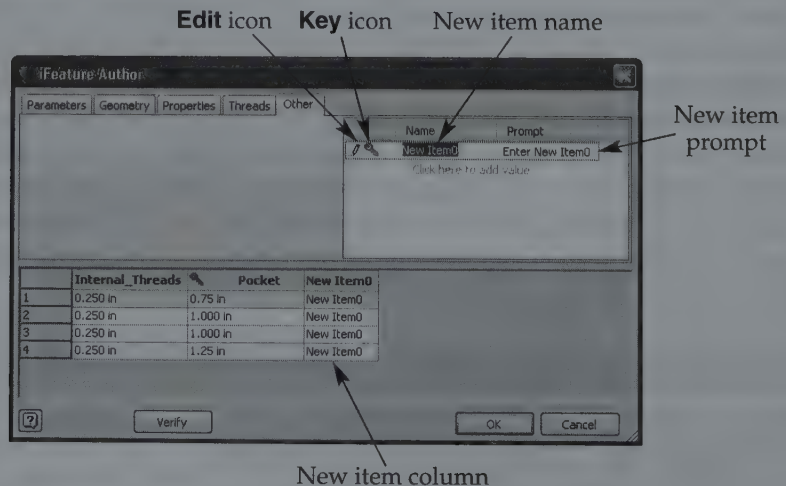
Figure 14-14.
Creating an additional iFeature table row.

Multiple rows added and cell values changed to represent each variation
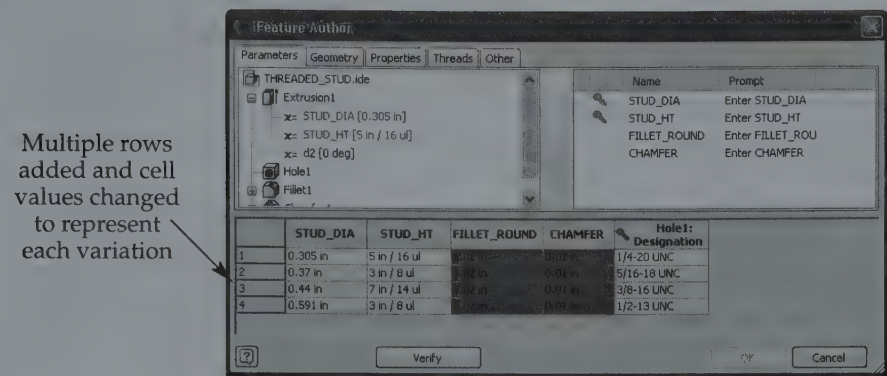
creating multiple variations of a threaded stud. You can enter any value in table cells, but suppression values require certain expressions. To specify a feature as suppressed, so that the feature "turns off" during insertion, enter Suppress, S, s, OFF, Off, off, or O. To identify a feature as computed, enter Compute, U, u, C, c, ON, On, on, or 1. Continue adding rows and defining cell values as required to generating the desired number of iFeature variations. To delete a row, right-click on the row and select **Delete Row**.

> **NOTE**
>
> If you do not specify a unit of measurement in a table cell, the default document units apply. To define a row as the default iFeature variation, right-click and select **Set as Default Row**.

When you specify a value in a cell, such as 3 or 5, the value is static. Custom columns or cells remove this limitation. Use a custom column to specify a value not listed in a cell. To create a custom column, right-click on the column or the column name in the **Selected Items** list, and pick **Custom Parameter Column**. Use a custom cell to specify a value not listed in a cell, but only if the value in another column does not control the cell. To create a custom cell, right-click on the cell and pick **Custom Parameter Cell**.

You can also set custom columns and cells to allow an *increment* or to allow values within a range. Right-click on a custom column or cell and choose **Specify Range For Column...**, **Specify Increment For Column...**, **Specify Range For Cell...**, or **Specify Increment For Cell...** to set the range or increment using the **Specify Range or Increment** dialog box.

increment: A set amount by which values increase in equal steps. For example, with an increment of 2, a size would increase to 4, 6, 8, 10, and so on.

> **NOTE**
>
> You cannot specify a column or cell range or increment for certain items, including some properties, position geometry, and threads. Custom columns and cells display blue backgrounds.

Finalizing the iFeature

After you create a table in the **iFeature Author** dialog box, pick the **Verify** button to check each record in the table to make sure they are all unique. If no errors are found, pick the **OK** button to generate the table. The final step in creating a table-driven iFeature is to resave the iFeature file.

The browser includes a number of options related to working with iFeatures, especially table-driven iFeatures. Expand the iFeature and the iFeature table to display rows. For example, **Figure 14-15** shows a table-driven iFeature with two rows. Each row includes *only* keyed columns. The check boxes next to column items indicate the iFeature currently displayed in the graphics window. To observe a different iFeature variation, double-click on the last attribute. For example, in **Figure 14-15**, double-click Fillets = 0.25 in. to observe the 2″ iFeature, or double-click Fillets = 0.125 in. to observe the 1″ iFeature.

To edit the table using the **iFeature Author** dialog box, access the **iFeature Author Table** tool, right-click on the table in the browser, and select **Edit Table...**, or double-click the iFeature table. You can also edit an iFeature table using the embedded Microsoft® Excel spreadsheet. See **Figure 14-16**. Access the spreadsheet using the **Edit Using Spread Sheet** tool, or right-click on the table in the browser and select **Edit via Spread Sheet...**.

Exercise 14-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 14-5.

Figure 14-15.
An example of the browser display for a table-driven iFeature.
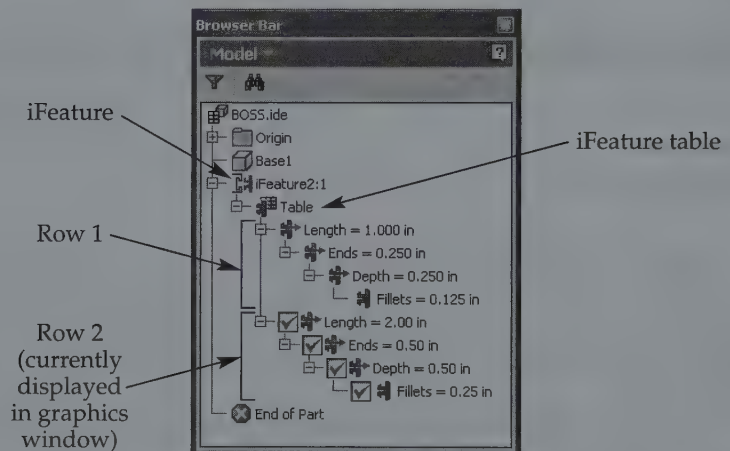
Figure 14-16.
An example of an embedded Microsoft® Excel spreadsheet created when you generate a table-driven iFeature.

Access the **Insert iFeature** tool to insert a table-driven iFeature using the **Insert iFeature** dialog box. The process of selecting and positioning a table-driven iFeature is the same as for a standard iFeature. Pick the **Browse...** button to use the **Open** dialog box to locate an iFeature anywhere on your computer or the network. The **Position** area is active and a preview of the iFeature appears in the graphics window. See **Figure 14-17**.

The difference between inserting a standard and table-driven iFeature is noticeable when you pick the **Next** button or select **Size** on the tree structure to activate the **Size** area. See **Figure 14-18**. Select the default value in the **Value** list to choose a different column value using the **Select Value** list box. To show all available values, pick the **All Values** check box. The value you select determines the row used to construct the iFeature. For example, picking the 1/2-13 UNC value shown in **Figure 14-18** tells Inventor to use all of the column values in the row where the 1/2-13 UNC thread is found to create the appropriate feature. To exit the **Select Value** list box without choosing a value, press the [Esc] key. A custom column does not include a key, allowing you to edit the value like any standard iFeature, by entering or selecting a new value.

Pick the **Next** button or select the **Precise Pos.** option on the tree structure to activate the **Precise Position** area. Select the appropriate radio button to specify whether you want to edit the iFeature sketch. Pick the **Finish** button to place the iFeature and, if specified, activate the sketch.

Ribbon
iFeature
> iFeature

Edit Using Spread Sheet

Ribbon
Manage
> Insert
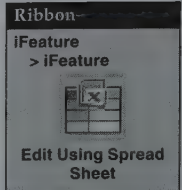
Insert iFeature

NOTE

Edit table-driven iFeatures using the same standard iFeature editing techniques previously described.
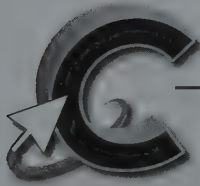
Exercise 14-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 14-6.

Figure 14-17.
The **Insert iFeature** dialog box that appears when you select a table-driven iFeature to insert. Notice the **Table Driven** icon.

Table Driven icon

Pick to flip direction

Specify a rotation angle

Preview of default variation

Selected reference plane

Pick the **Move** icon to relocate

Pick the **Rotate** icon to rotate
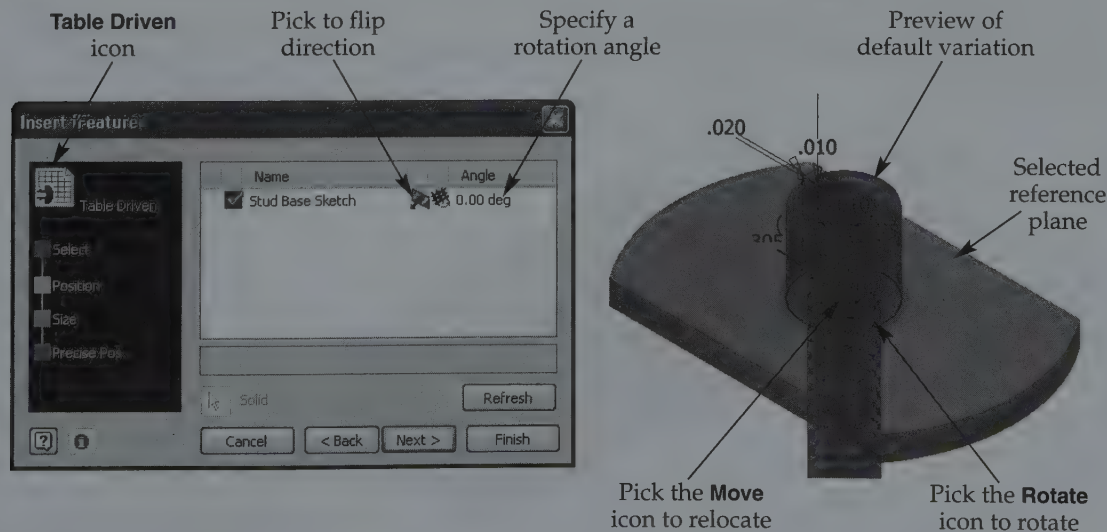
Figure 14-18.
Using the **Size** area of the **Insert iFeature** dialog box to size a table-driven iFeature.

Modify custom values as you would a standard iFeature size parameter

Select a value to change the iFeature from the default configuration
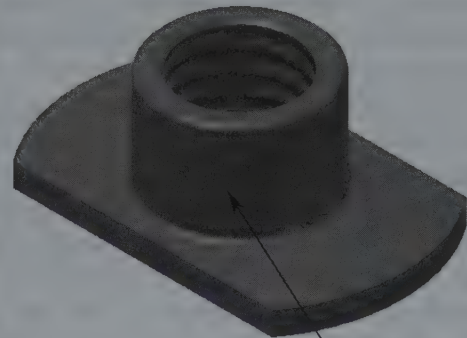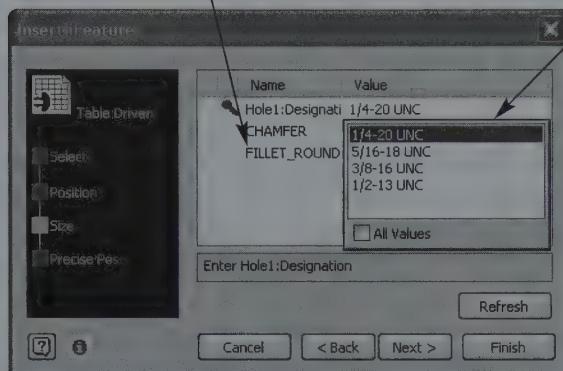
Selected value (row) controls all iFeature characteristics

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. What are catalog features?
2. What is the file extension for an iFeature?
3. Explain how to modify the default parameter names in an iFeature so that they reflect the dimension they control.
4. Where are iFeatures stored on the computer?
5. Briefly describe the process of creating an iFeature.
6. What tool allows you to place iFeatures in a part (.ipt) file?
7. Briefly explain how to edit an iFeature.
8. Which tool allows you to view and manage iFeatures that are stored in the default location?
9. What is a table-driven iFeature and how does it work?
10. Explain the disadvantage of having only a single row in an iFeature table.

Problems

Instructions:

- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. Perform the following tasks:

    A. Begin a new part file using the Part-IN.ipt or Part-mm.ipt template from the **New File** dialog box.

    B. Access the **Application Options...** dialog box, explore the **iFeature** tab, and modify the settings if needed. Close the **Application Options** dialog box.

    C. Access the **View Catalog** tool and explore the default Catalog folder.

    D. Pick the Geometric Shapes folder and open the Cone.ide file. Close the file.

    E. Close the iFeature viewer and close the part file without saving.

▼ Basic

2. **Title:** DIE

    Units: Inch

    Template: Part-IN.ipt

    Part Number: IAA-102-01

    Project: TRAY

    Material: Steel, High Strength Low Alloy

    Color: Default

    Save as: P14-2.ipt

    Specific Instructions: Open a sketch on the XY plane and sketch a 2″ × 3″ rectangle around the projected center point. Extrude the sketch 1″ in the positive direction. Access the **Insert iFeature** tool and use the following information to insert the iFeature: C:\Program Files\Autodesk\Inventor 2010\Catalog\Pockets and bosses\Boss_obround_2_fillets.ide.

    A. Pick the top extrusion face as the Sketch Plane and rotate the iFeature 90°.

    B. Specify the length as 2.5″, but do not change any other values.

    C. Select the **Activate Sketch Edit Immediately** radio button and add the dimensional constraints shown.

    The final part should look like the part shown.
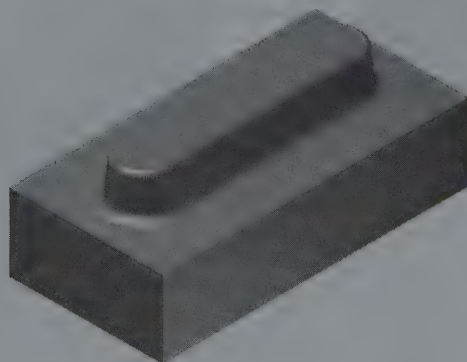
3. **Title:** FLUSH BRACKET
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-103-01
   Project: INSTRUMENT MOUNT
   Material: Steel
   Color: Default
   Save as: P14-3.ipt
   Specific Instructions: Access the **Insert iFeature** tool and use the following information to insert the ANGLE iFeature you created during Exercise 14-1.
   A. Pick the center point in the **Origin** folder of the browser as the Origin and the XY plane as the Sketch Plane. Do not move or change the direction or angle.
   B. Specify the width as 1″, height as 1.5″, thickness as .125″, and length as 1.5″.
   C. Select the **Do not Activate Sketch Edit** radio button.
   First Stud: Access the **Insert iFeature** tool and use the following information to insert the THREADED_STUD table-driven iFeature you created during Exercise 14-5.
   A. Pick the plane shown as the Stud Base Sketch. Do not move or change the direction or angle.
   B. Specify the 1/4-20 UNC hole designation, but do not change the fillet, round, and chamfer custom values. Select the **Activate Sketch Edit Immediately** radio button and add the concentric constraint shown.
   Second Stud: Access the **Insert iFeature** tool and use the following information to insert a second copy of the THREADED_STUD table-driven iFeature.
   A. Pick the plane shown as the Stud Base Sketch. Do not move or change the direction or angle.
   B. Specify the 3/8-16 UNC hole designation, but do not change the fillet, round, and chamfer custom values. Select the **Activate Sketch Edit Immediately** radio button and add the concentric constraint shown.
   The final part should look like the part shown.

Vertical   Vertical   Horizontal   .440   Horizontal   .305   Horizontal

First Stud   **Second Stud**   **Final Part**
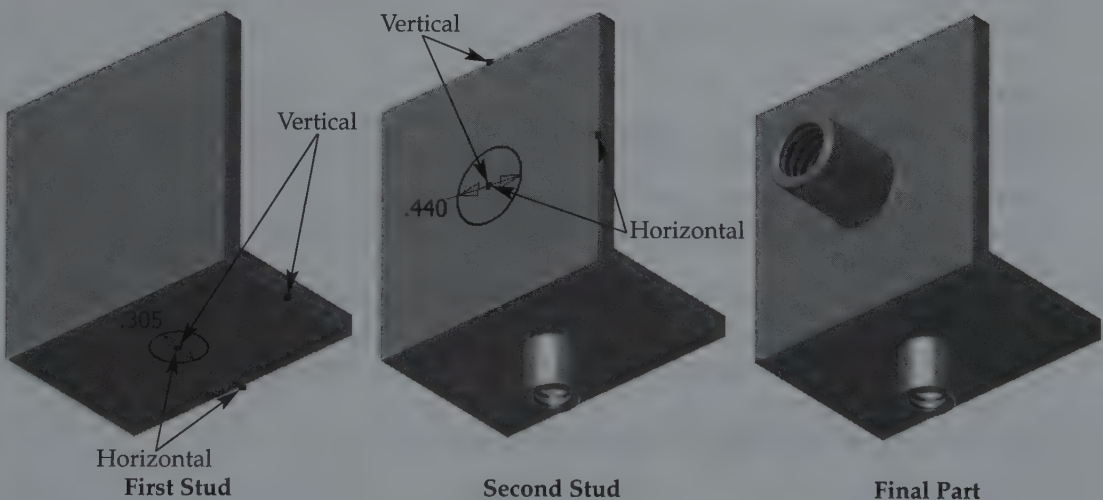
4. **Title:** BEARING POCKET
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-034-01
   Project: IFEATURE
   Material: Default
   Color: Default
   Save as: P14-4A.ipt
   Specific Instructions: Create a 1" × 2" × 2" extruded rectangle. Open a sketch on the top face of the extrusion, and place the hole centers as shown. Create the two smaller holes as specified. Share Sketch2. Create the larger middle hole as specified. Use the **Parameters** dialog box to rename the internal thread diameter (Hole1) to Internal_Threads, and the pocket diameter (Hole2) to Pocket. Create an iFeature using the following information:
   A. Name: Bearing_Pocket
   B. Selected features: Hole1 and Hole2
   C. Limit the value of Internal_Threads to a range of .125 < .25 < .5
   D. Change the prompt for Internal_Threads to Enter Diameter of Internal Threads
   E. Limit the value of Pocket to a list of .75", 1", and 1.25"
   F. Change the prompt for Pocket to Enter Pocket Diameter
   G. Save Bearing_Pocket as P14-1.ide.
   Access the **View Catalog** tool and open P14-1.ide. The iFeature should look like the feature shown.
   Convert the iFeature to a table-driven iFeature and develop a table of six different bearing pocket designs. Save and close the iFeature file. Create a new part of your own design using each different variation of the bearing pocket. Save the part as P14-4B.

2X 1/4-20 UNC - 2B

Ø1.00 ▼ 0.50

1.750

1.000

.250

1.000
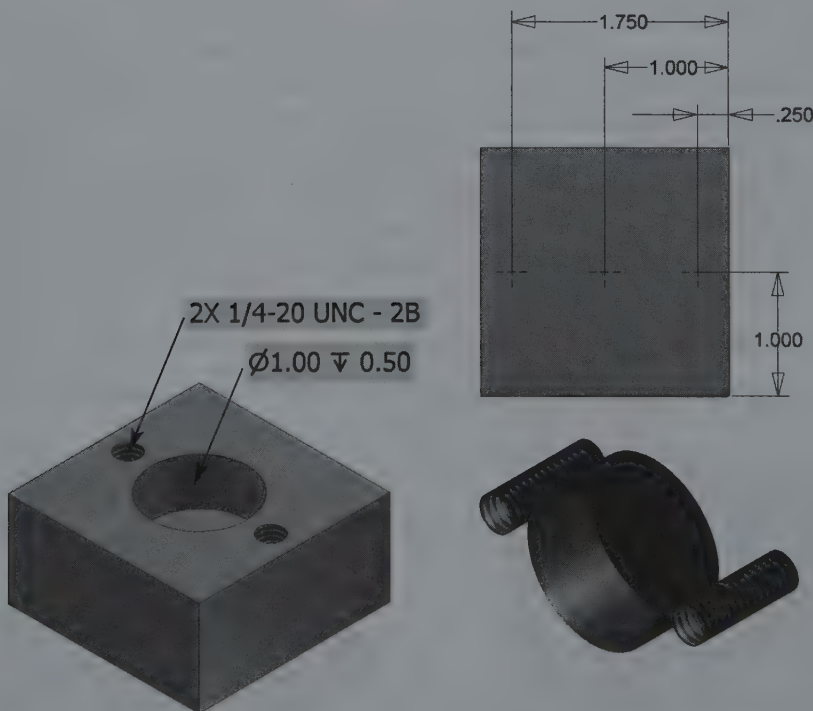
5. **Title:** BEARING POCKET BASE
   Units: Inch
   Template: Part-IN.ipt
   Part Number: IAA-035-01
   Project: IFEATURE
   Material: Default
   Color: Default
   Save as: P14-2.ipt
   Specific Instructions: Create a 1″ × 3″ × 3″ extruded rectangle. Insert the P14-1.ide iFeature on the top face of the extrusion. Attempt to enter a value of .5″ in the **Internal_Threads Value** text box. Select the **Activate Sketch Edit Immediately** radio button. Using sketch dimensions, center the iFeature in the middle of the extrusion, and finish the sketch. Right-click Bearing_Pocket1 in the browser and select **Edit iFeature....** Change the bearing pocket diameter to .75″. Place a .25″ chamfer around the top edge of the extrusion. The final part should look like the part shown.
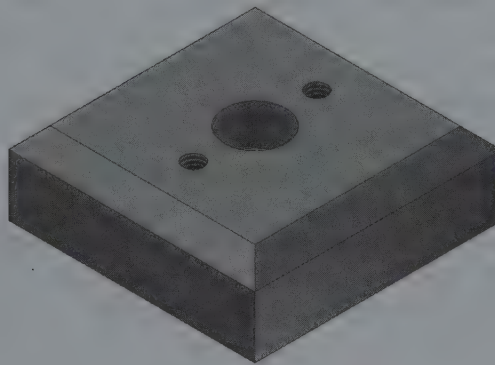
Introduction to Sheet Metal Parts

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Begin a new sheet metal part.
✓ Convert a part to a sheet metal part.
✓ Use and modify sheet metal styles.
✓ Create faces and contour flanges.
✓ Add flanges and hems.
✓ Create lofted flanges and contour rolls.

Sheet metal products are common in the automotive, electronic, product chassis, and heating, ventilating, and air conditioning (HVAC) industries. Inventor includes specific tools and options for modeling *sheet metal parts*. See **Figure 15-1**. Manufacturers input model data into software that controls sheet metal part production and transfers a *flat pattern*, as shown in **Figure 15-1B**, to a sheet of metal. The metal is then *cut*, drawn, stretched, and *bent* to the correct specifications. You can use sheet metal model tools for any application or material that requires similar forming and flat pattern processes, such as cardboard package design.

sheet metal parts: Parts formed from a flat sheet of metal.

flat pattern: A 2D drawing representing the final, unfolded part.

cut: Any process, such as shearing, punching, or laser, water jet, or similar process that is used to remove material.

bent: Formed using a stake, brake, folder, die, roller, or similar tools.

Figure 15-1.
A—An example of a formed sheet metal part model. B—The flat pattern of the model shown in A.

A   B

Sheet Metal Model Fundamentals

Modeling a sheet metal part is similar to modeling a non-sheet metal part. Usually you begin with a 2D or 3D sketch, although a 2D sketch is most common. Use the same sketching tools and techniques you use to prepare a non-sheet metal part sketch. You then create a sheet metal sketched base feature, and add sketched, placed, and catalog features as needed. Most often, you build a sheet metal part in a finished, folded form that you can unfold to create a flat pattern.

A sheet metal part is formed from a single piece of material according to parameters appropriate for the material, part, and manufacturing process. As a result, sheet metal models reference several specifications in addition to color and lighting. These parameters, or rules are set by styles and include metal thickness, material, bend radii, and relief sizes. For most applications, sheet metal rules remain the same throughout part construction and are automatically applied as you design. This technique replicates real-world sheet metal part manufacturing by forming parts using a specific type of sheet metal and sheet metal rules. Sheet metal rules also control how flat patterns appear.

NOTE

The sheet metal work environment provides many standard modeling tools, including **Hole**, work feature, feature pattern, parameter, and iFeature tools. These tools behave the same as in the standard part model environment.

PROFESSIONAL TIP

Remember when modeling a sheet metal part that material thickness and several other sheet metal parameters are predefined. Open profile sketches are common.

Sheet Metal Templates

Both non-sheet metal and sheet metal part files carry the file extension .ipt. Typically, when you begin a new design, you specify whether the part is for a non-sheet metal or sheet metal part application. Use a sheet metal part template to create a sheet metal part using sheet metal tools and options.

Converting Parts to Sheet Metal Parts

If you begin modeling from a non-sheet metal part template, you must convert to the sheet metal part environment to access sheet metal tools. Access the **Convert to Sheet Metal** tool to change from a modeling design to a sheet metal design. You can also convert from a sheet metal design to a modeling design.

Ribbon
Model
> Convert

Convert to Sheet
Metal

CAUTION

The **Sheet Metal** environment includes specialized sheet metal part model tools, such as **Face**, that reference a preset material thickness and sheet metal parameters. An alert notifies you of this condition when you convert a model. You can access non-sheet metal tools, such as **Extrude**, from within the sheet metal work environment. However, for most applications, you should not use non-sheet metal and sheet metal part features together.

Exercise 15-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-1.

Exercise 15-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-2.

Sheet Metal Styles

A sheet metal part references the color, lighting, sheet, bend, corner, and unfold settings specified in the **Style and Standard Editor**. Use sheet metal styles appropriate for the material, part, and manufacturing process. For example, use a style assigned .0478" (18 gage) steel attributes for an 18 gage steel part. If the design changes, requiring the part to be .0453" (17 gage) aluminum, for example, change the part to reference a style with those attributes. The part automatically updates according to the new parameters.

The **Sheet Metal Defaults** dialog box, shown in **Figure 15-2**, provides a convenient way to select a sheet metal style and override sheet metal style settings. Use the **Sheet Metal Rule** drop-down list to activate a *sheet metal rule*. By default, all other options in the **Sheet Metal Defaults** dialog box reference values specified by the active sheet metal rule. This is suitable for most applications.

Assigning a thickness, material style, or unfold rule different from the sheet metal rule overrides the style in the current document. To edit or create a sheet metal, material, or unfold rule, access the **Style and Standard Editor**. Pick the appropriate button in the **Sheet Metal Defaults** dialog box to open the **Style and Standard Editor** and display the corresponding node, such as the **Sheet Metal Rule** node shown in **Figure 15-3**.

sheet metal rule: The style that controls all sheet metal part sheet, bend, and corner settings and specifies the unfold style.

Ribbon

Sheet Metal > Setup

Sheet Metal Defaults

Ribbon

Manage > Styles and Standards

Styles Editor
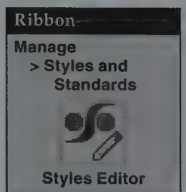
Figure 15-2.
Use the **Sheet Metal Defaults** dialog box to select a sheet metal rule or override sheet metal rule parameters for the current design session.

Select sheet metal rule

Deselect to override material thickness using the **Thickness** text box

Change to override the material style

Change to override the unfold rule

Pick to select, create, or modify the sheet metal rule using the **Style and Standard Editor**

Pick to select, create, or modify the material style using the **Style and Standard Editor**

Pick to select, create, or modify the unfold using the **Style and Standard Editor**

Figure 15-3.
Use the **Style and Standard Editor** to activate, create, and modify sheet metal rules. The **Sheet** tab of the **Sheet Metal** area controls sheet and flat pattern parameters.

Double-click to activate; active rule is bold

Pick to access the **Sheet Metal Unfold** area

Pick to access the **Material Style** area

Pick to access the **Sheet Metal Unfold** area
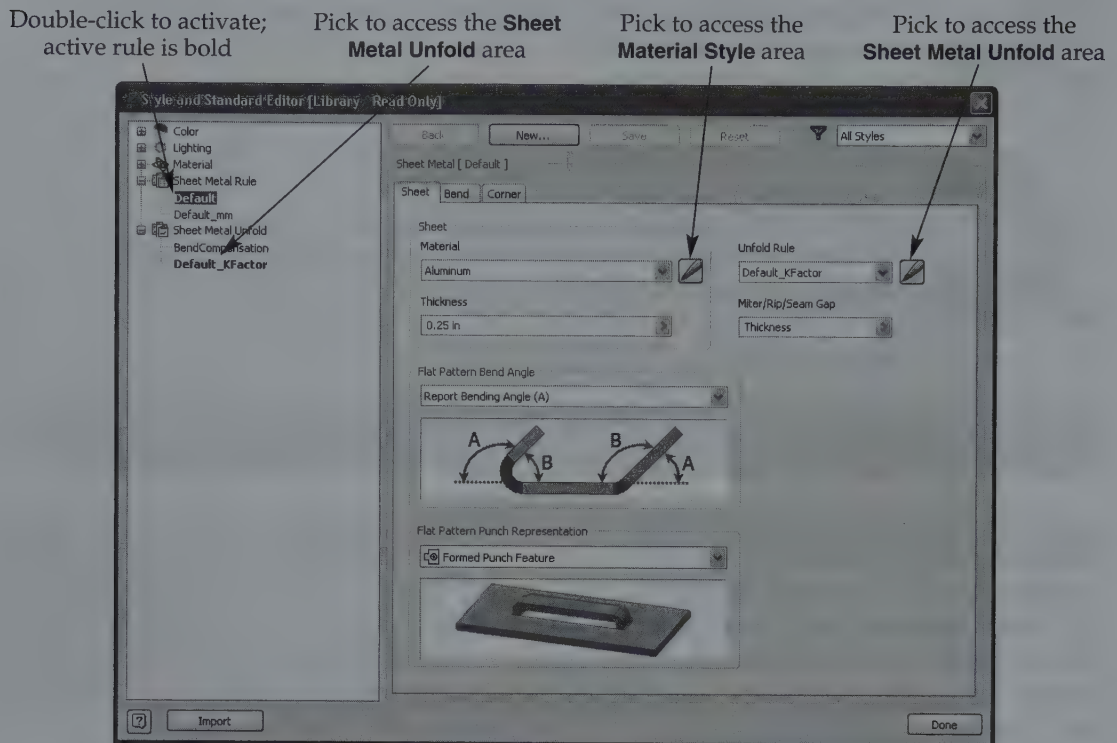
Sheet Metal Rules

Expand the **Sheet Metal Rule** node to display available sheet metal rules. Double-click on a sheet metal rule or right-click and select **Active** to activate it. Pick a sheet metal rule to display tabs with sheet metal rule options. Create a sheet metal rule for each unique material, part, or manufacturing requirement.

> **NOTE**
>
> You can override many sheet metal rule settings for unique requirements while you are creating sheet metal features.

Sheet Options

The **Sheet** tab, shown in **Figure 15-3**, controls sheet and flat pattern parameters. Specify the material type using the **Material** drop-down list, and specify material thickness using the **Thickness** text box. The material set in the **Physical** tab of the **iProperties** dialog box adapts to the current sheet metal rule material. Changing the file material overrides the sheet metal rule.

Use the **Unfolding Rule** drop-down list to select a predefined sheet metal unfold rule. Unfold rules and settings are available in the **Sheet Metal Unfold** node of the **Style and Standard Editor**, described later in this chapter. For a sheet metal model to unfold, the model must include acceptable *seams*. Specify a value in the **Miter/Rip/Seam Gap** text box to define the default seam opening. See **Figure 15-4**.

seam: The line or opening formed when the ends of the pattern come together.

Use the **Flat Pattern Bend Angle** drop-down list to specify the included angle referenced by the flat pattern. Choose **Report Bending Angle (A)** option to callout the bend angle, such as BEND 45°, or select the **Report Open Angle (B)** option to callout the open angle, such as OPEN 135°. Pick an option from the **Flat Pattern Punch Representation** drop-down list to define how sheet metal punches created using the **PunchTool**

Figure 15-4.
The **Miter/Rip/Seam Gap** value sets the default space between seam edges for sheet metal modeling tools. Notice how the Thickness parameter and an equation relate the gap size to the material thickness.
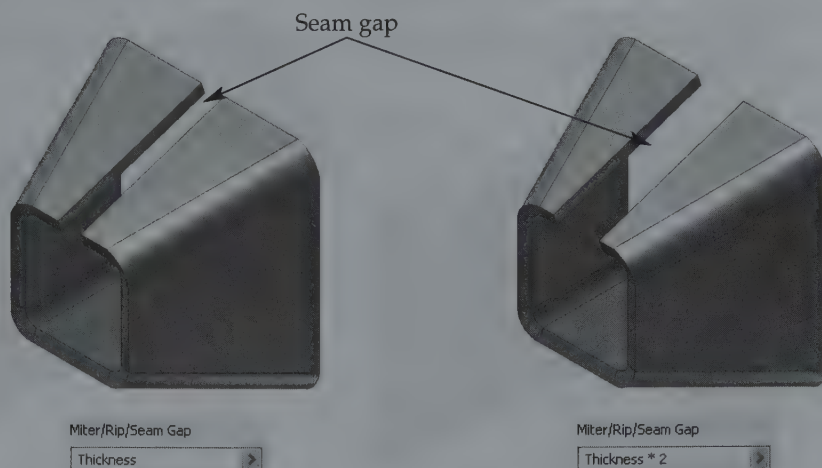
appear on the flat pattern. View the preview of each option to pick the appropriate representation.

Bend Options

The **Bend** tab, shown in **Figure 15-5**, controls default bend rules. Use the **Relief Shape** drop-down list to select a *bend relief* shape. See **Figure 15-6**. The **Relief Width** text box allows you to adjust the relief width using a value greater than 0. See **Figure 15-7A**. The **Relief Depth** text box allows you to adjust the relief depth using a value greater than or equal to 0. See **Figure 15-7B**. Use the **Minimum Remnant** text box to specify whether material remains after a bend operation, based on the size of the leftover material. The leftover material is removed if the amount of material is less than the minimum remnant value. See **Figure 15-8**.

bend relief: Relief typically added to a sheet metal part to relieve stress, or the tear, that occurs when a portion of a piece of material is bent.

Figure 15-5.
The **Bend** tab of the **Sheet Metal** area. Notice the default values set in reference to the material thickness parameter.
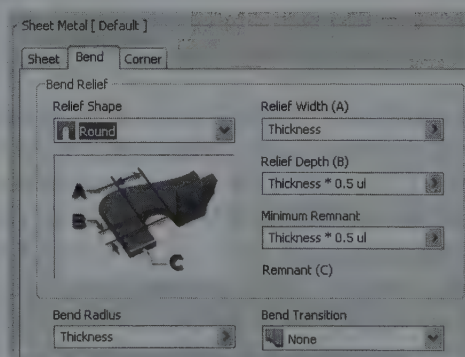
Figure 15-6.
Examples of straight relief, round relief, and tearing at the bend to apply no relief.

Straight   Round   Tear
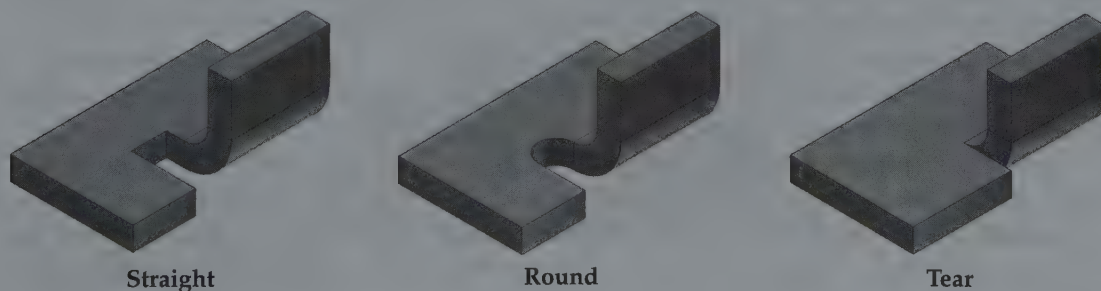
Figure 15-7.
A—An example of a relief width greater than the material thickness. B—An example of a relief depth greater than 0.
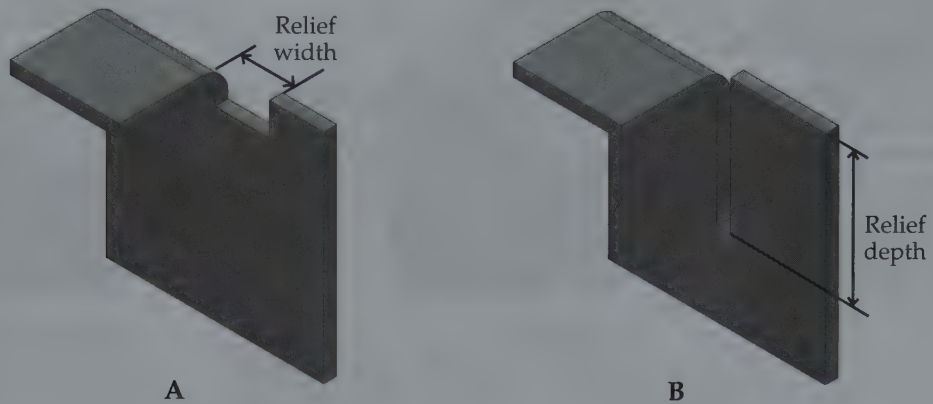
Figure 15-8.
The minimum remnant value must be greater than the size of the remnant material created by a bend in order for the material to be removed.
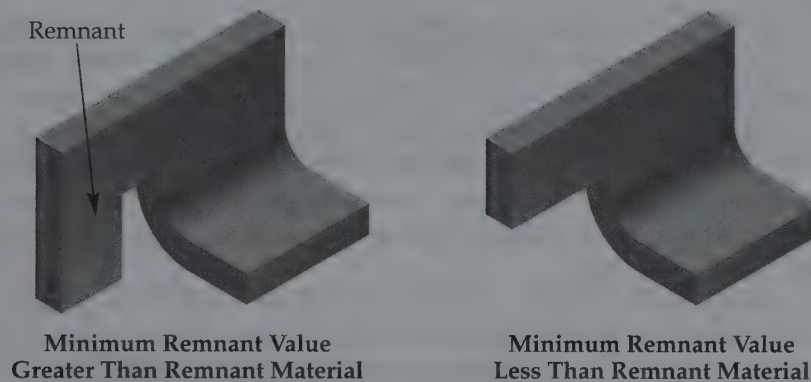
bend radius: The inside radius of a formed feature.

Specify a default *bend radius* greater than 0 using the **Bend Radius** text box. Select the default bend transition type from the **Transition** drop-down list. The transition setting is especially important for bending across complex shapes, to ensure manufacturability. Except for the **Trim to Bend** option, all transition types appear in the folded model as if the **None** option is active. The flat pattern shows the effects of selecting an option other than **None** or **Trim to Bend**. See **Figure 15-9**. The **Transition Radius** text box appears when you pick the **Arc** option, allowing you to specify the arc transition radius.

Corner Options

corner relief: Relief typically added to a sheet metal part to relieve stress at a bend corner at the intersection of two or three faces.

The **Corner** tab, shown in **Figure 15-10**, controls default *corner relief* parameters. Use the **2 Bend Intersection** area to adjust relief at the intersection of two bends. Select a relief shape from the **Relief Shape** drop-down list. The **Relief Size** text box is enabled when you select the **Round** or **Square** option, allowing you to set the diameter of a round relief or the length and width of a square relief. **Figure 15-11** shows an example of each **2 Bend Intersection** relief option.

Use the **3 Bend Intersection** area to adjust relief at the intersection of three bends. Select a relief shape from the **Relief Shape** drop-down list. The **Relief Radius** text box is enabled when you select the **Round with Radius** option, allowing you to set the radius of a relief arc. **Figure 15-12** shows an example of each **3 Bend Intersection** relief option.

Figure 15-9.
Select a transition type based on how you want bends to form between features. Bend transition is influenced by all bend factors, including the bend position.

None

Spline from bend edge

Intersection

Straight line at intersection

Straight Line

Straight line from bend edge

Arc

Specified arc radius

Folded Model

Trim to Bend

Trim to bend

Transition begins

Trim appears
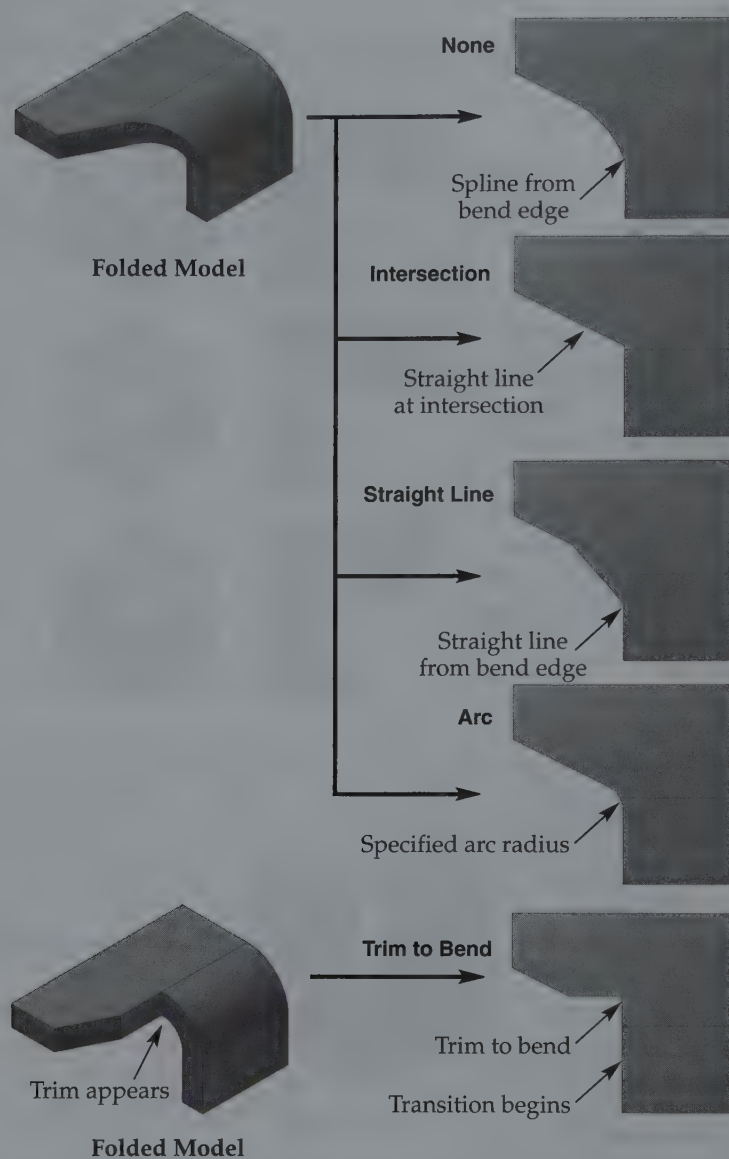
Folded Model

Figure 15-10.
The **Corner** tab of the **Sheet Metal** area.

Sheet Metal [Default]

Sheet | Bend | Corner

2 Bend Intersection
Relief Shape
Trim to Bend

Relief Size

Select a relief shape at the intersection of two bends

3 Bend Intersection
Relief Shape
Round with Radius

Relief Radius
BendRadius

Select a relief shape at the intersection of three bends

Round

Trim to Bend

Square

Linear Weld

Tear (no relief)

Arc Weld
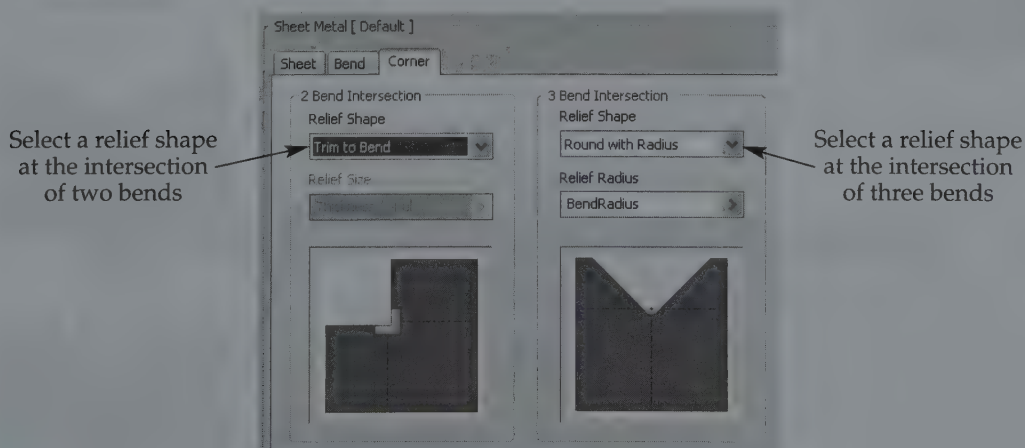
Figure 15-12.
Corner relief shapes applied at the intersection of three bends. In the folded model, all relief shapes appear as if the **None** option were active. The flat pattern shows the actual effects of selecting an option other than **None**.

Folded Model

Original model geometry

No Replacement

Intersection

Full Round

Specified radius

Round with Radius
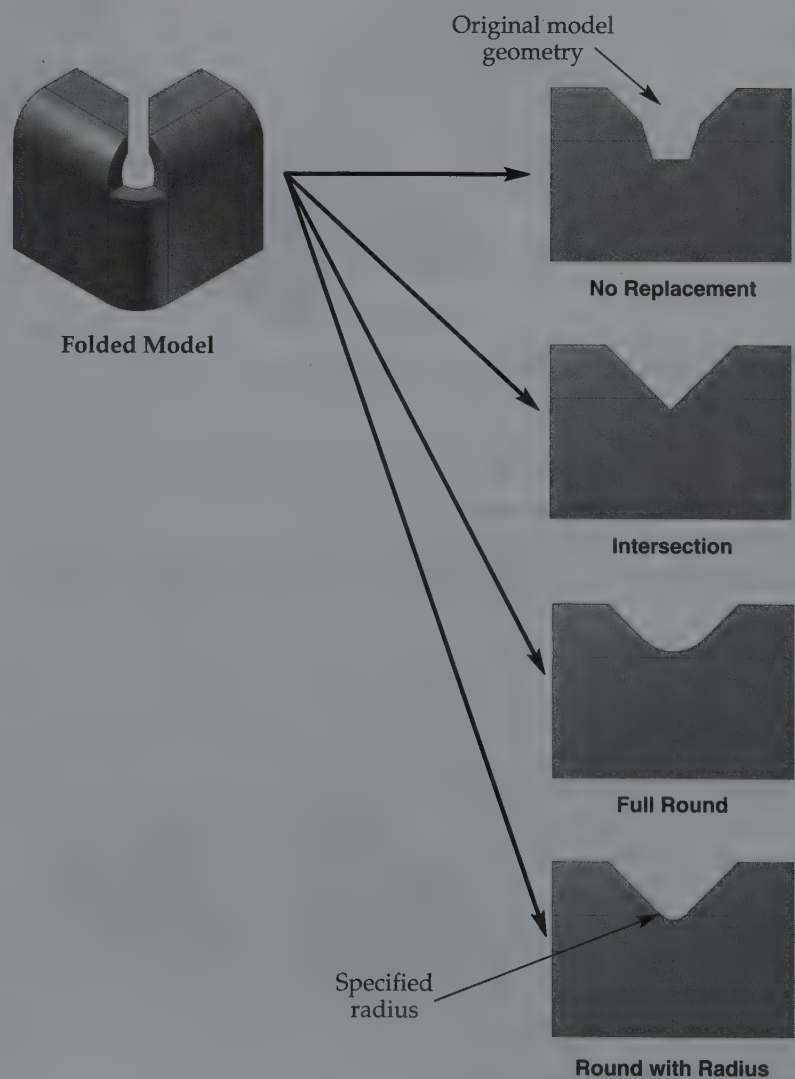
Sheet Metal Unfold Rules

Expand the **Sheet Metal Unfold** node to display available sheet metal unfold rules. Double-click on an unfold rule or right-click and select **Active** to activate it. Pick an unfold rule to display options. In general, the unfold rule is a set of parameters that determines the *bend allowance*.

Bend allowance calculations determine the overall dimension of the flat pattern, needed to achieve the correct folded dimension when the part is bent. Several methods and formulas are available for establishing bend allowance, by inputting multiple sheet metal variables. Companies often determine bend allowance and prepare bend allowance charts for various materials through experimentation. In general, each material thickness receives a bend deduction calculation, based on the specified bend angle, bend radius, and k-factor.

Select **Linear** from the **Unfold Method** drop-down list to specify a *k-factor* to locate the *neutral axis* in the **KFactor Value** text box. See **Figure 15-13**. A second option is to use a bend table by picking the **Bend Table** option from the **Unfold Method** drop-down list. The third option is to pick the **Custom Equation** option from the **Unfold Method** drop-down list. This option allows you to create a custom set of bend calculations based on bend allowance, compensation, deduction, or k-factor.

bend allowance: The amount of extra material needed for a bend to compensate for compression during the bending process.

k-factor: A multiple, typically between .25 and .5, that locates the neutral axis.

neutral axis: The axis of a bend radius where neither stretching nor compressing occurs.

Creating and Saving Sheet Metal Styles

To create a new sheet metal rule or unfold rule, select a style with characteristics similar to the new style, and then pick the **New** button to create a copy of the style. Enter a style name in the **New Style Name** dialog box. If changes you make to a style are not appropriate, pick the **Reset** button to return to previous style settings *before* saving the style. Pick the **Save** button to save changes to a style to the current document. If you do not save after making changes, a prompt asks you to do so before continuing.

Save the style to a template and reference it each time you create a new model using the template, or save it to the *style library*. The style library provides a method of storing all style information in a location other than individual Inventor files. Styles are copied from the style library to files as needed for use in existing and future models. You can update any changes made to a style saved in the style library so that the style is correct in all the files that reference the style.

style library: A folder, Design Data by default, that houses styles in XML file format.

> **NOTE**
>
> Pick the **Import** button to import existing sheet metal and unfold rules, or bend tables in a .txt file format. Right-click on a sheet metal or unfold rule and pick **Export...** to export it, or export a completed bend table as a .txt file by picking the **Export Table** button.

Figure 15-13.
Select the **Linear** option to enter a k-factor.

Active rule

Pick to select a linear or bend table unfold method

Exercise 15-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-3.

Exercise 15-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-4.

Faces

sheet metal face: A feature that uses a closed profile sketch to create a planar sheet metal surface.

Ribbon

Sheet Metal > Create

Face

double bend: A bend between two parallel faces that are not coplanar.

A *sheet metal face* is a common base feature. Access the **Face** tool to create a face using the **Face** dialog box. See **Figure 15-14**. If a sketch includes one profile, the profile is selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick profile(s). If necessary, pick the **Offset** button to flip the side of the sketch plane from which thickness occurs.

Additional options for connecting features are available when you add a face. See **Figure 15-15**. Adjust the bend radius using the **Radius** text box only if you want to override the default bend radius, identified as BendRadius. The **Edges** button allows you to select edges to form a *double bend*. Pick the **More** button to access the **Double Bend** area, and select the type of double bend. See **Figure 15-16**. If the preview does not look correct, pick the **Flip Fixed Edge** button to change the edge from which the bend originates. **Figure 15-17** shows extend options for creating a full radius double bend. Pick the **Extend Bend Perpendicular to Side Faces** button in the **Bend** area to form a

Figure 15-14.
Use the **Shape** tab of the **Face** dialog box to select a sketch profile and specify face parameters. The **Unfold Options** and **Bend** tabs allow you to override sheet metal rules applied to the face.
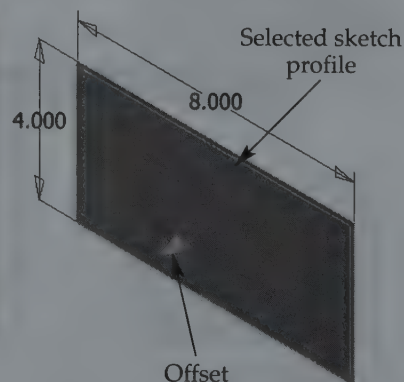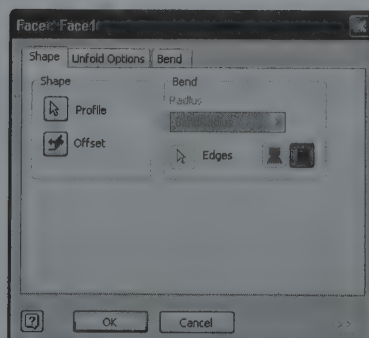
Figure 15-15.
Adding a sheet metal face. This example shows creating a double bend face between two parallel, non-coplanar sheet metal faces using the **Fix Edges** option. You do not need to add a double bend to create a secondary face.

Extend Bend Aligned to Side Faces button

Extend Bend Perpendicular to Side Faces button

Existing face

Selected double bend edge

Flip Fixed Edge button

4.000

Selected sketch profile on offset work plane
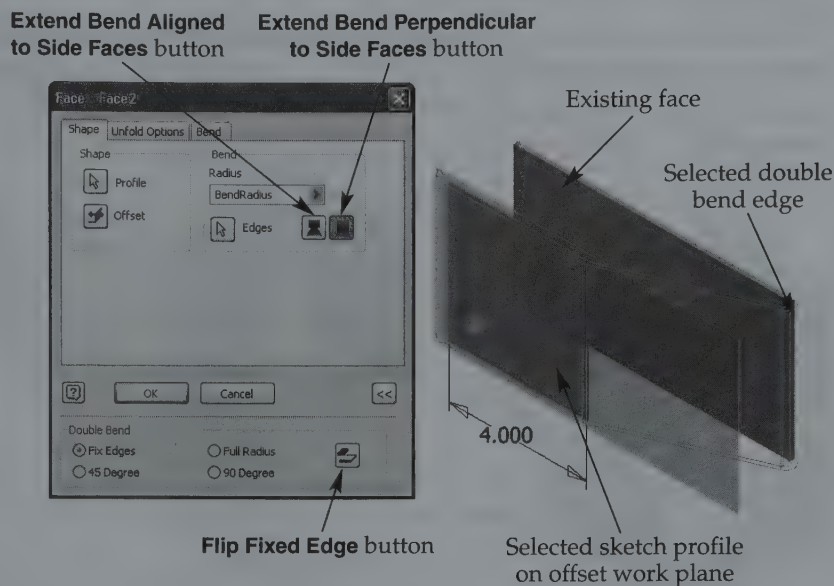
Figure 15-16.
Examples of double bend options.

Fix Edges

45 Degree

Full Radius

90 Degree
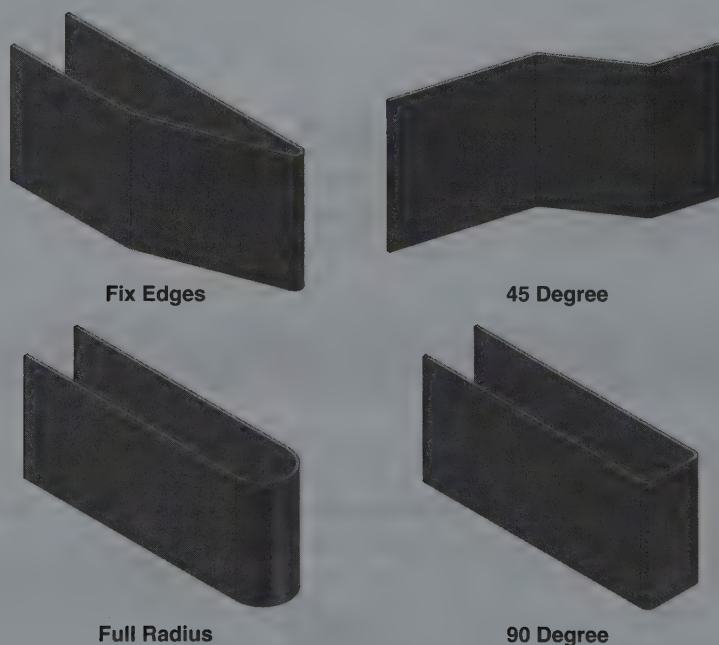
bend that extends perpendicular to side faces, as shown in **Figure 15-17B**. Pick the **Extend Bend Aligned to Side Faces** button to adjust the shape of the face by aligning the bend with profile geometry, as shown in **Figure 15-17C**.

The settings in the **Unfold Options** and **Bend** tabs are the same as those located in the **Style and Standard Editor**. Adjust the settings only if you want to override the default sheet metal parameters. Pick the **OK** button to create faces and exit, or pick the **Apply** button to create faces and remain in the tool.

PROFESSIONAL TIP

You can override specific bends by picking the **Bend Edit** glyph that appears when you add bends using some feature tools, or by expanding the feature in the browser, right-clicking on **Bend**, and selecting **Edit Bends**.

Figure 15-17.
A—An example of creating a full radius double bend face. B—Using the default **Extend Bend Perpendicular to Side Face** option. C—Applying the **Extend Bend Aligned to Side Faces** option.

Selected existing feature edge

Full radius bend preview

New face

Bend extends to perpendicular faces

Bend adjusts to align with side faces

A   B   C
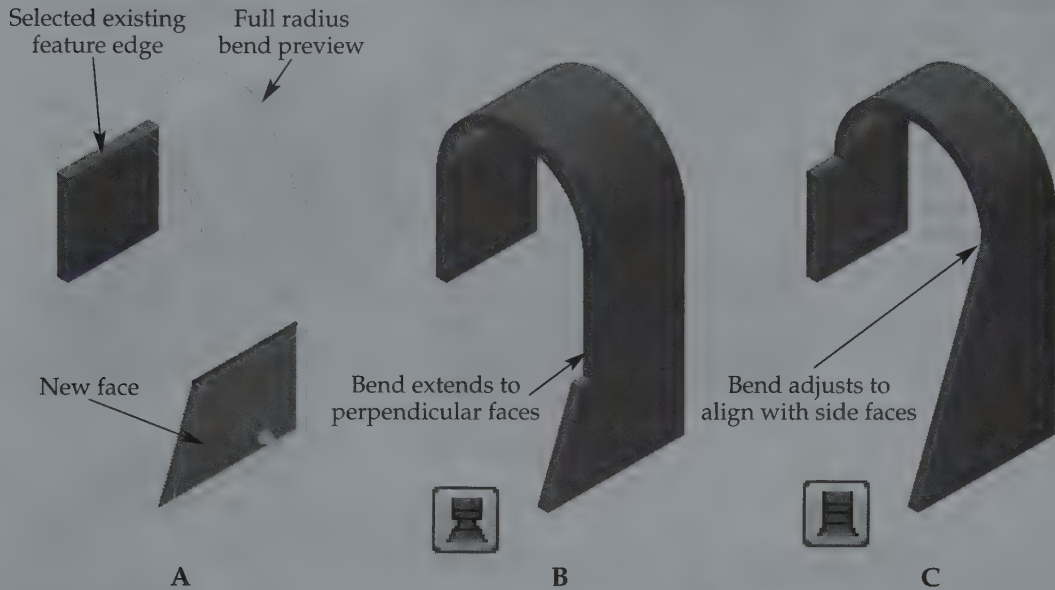
Exercise 15-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-5.

Contour Flanges

contour flange: A sheet metal feature that uses an open profile to create linear sheet metal faces.

Ribbon

Sheet Metal > Create

Contour Flange

A *contour flange* often replicates extruded sheet metal features such as gutters, corrugated panels, or a frame. Access the **Contour Flange** tool to create a contour flange using the **Contour Flange** dialog box. See **Figure 15-18**. The **Profile** button is active by default, allowing you to choose the sketch profile. Use the thickness flip buttons to control the side of the sketch profile from which thickness occurs. Pick the **Both Sides** button to divide the thickness equally on both sides of the profile.

A bend forms at the vertex of profile geometry. Do not sketch a radius at sharp corners. Adjust the bend radius using the **Radius** text box only if you want to override the default bend radius, identified as BendRadius. When creating a base feature, specify the contour flange length in the **Distance** text box. Use the distance flip buttons to control the side of the sketch plane from which flange distance occurs. Pick the **Distance Mid-plane** button to divide the distance equally on both sides of the plane.

Referencing Individual Edges

Additional options for connecting features are available when adding a contour flange. See **Figure 15-19**. To apply a contour flange along a single edge, select the sketch profile and then use the **Edge Select Mode** button to pick the edge. Next, pick the **More** button and select an option from the **Type** drop-down list. The **Edge** option is active by default and forms the contour flange along the entire length of the selected edge, as shown in **Figure 15-19**.

Figure 15-18.
Creating a contour flange using the **Contour Flange** dialog box.

Selected sketch profile

Preview

Use to control
the side of
the profile
where the
thickness
occurs

Use to control
the side of
the profile
where the
distance
occurs
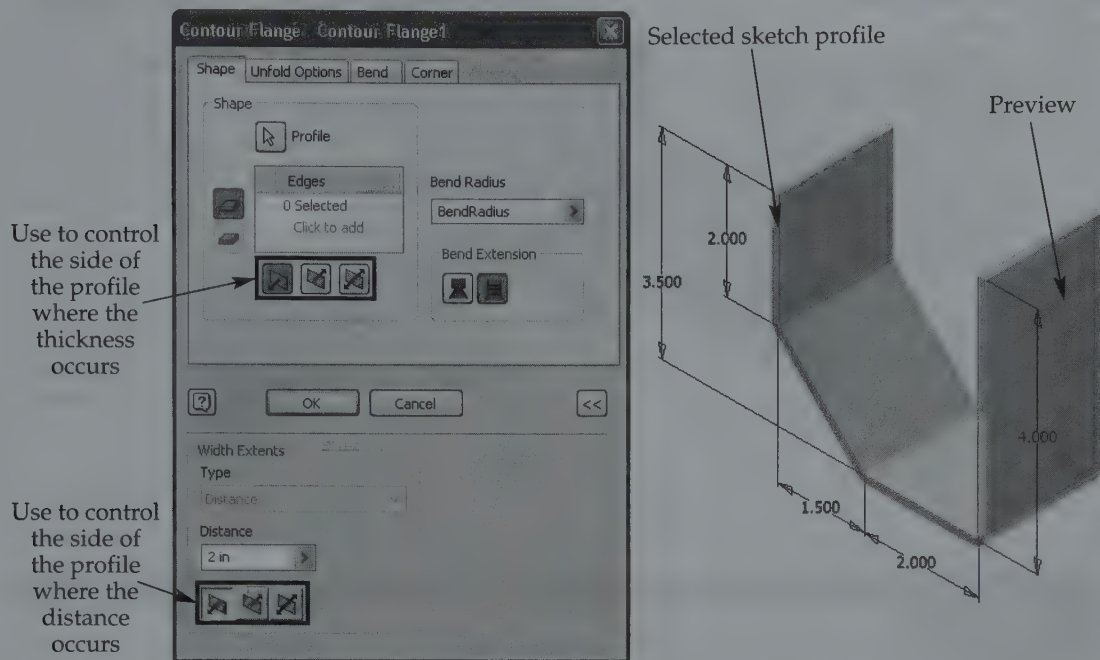
Pick the **Width** extents option to specify the contour flange width. The **Centered** radio button is active by default, centering the contour flange along the selected edge using the width specified in the **Width** text box. Instead of centering, pick the **Offset** radio button to define the width and an offset from a point on the edge. See **Figure 15-20**. The **Select Start Point** button becomes active, allowing you to pick an endpoint on the edge to form the offset. Specify an offset value using the **Offset** text box. If the offset occurs on the wrong side, pick the **Flip Direction** button. Specify the width of the contour flange using the **Width** text box.

Figure 15-19.
Adding a contour flange along a single edge using the **Edge** extents option.

Existing feature

Selected
feature edge

Pick to
select
individual
edges

Width
options
when a
single edge
is selected

Preview

Selected sketch
profile
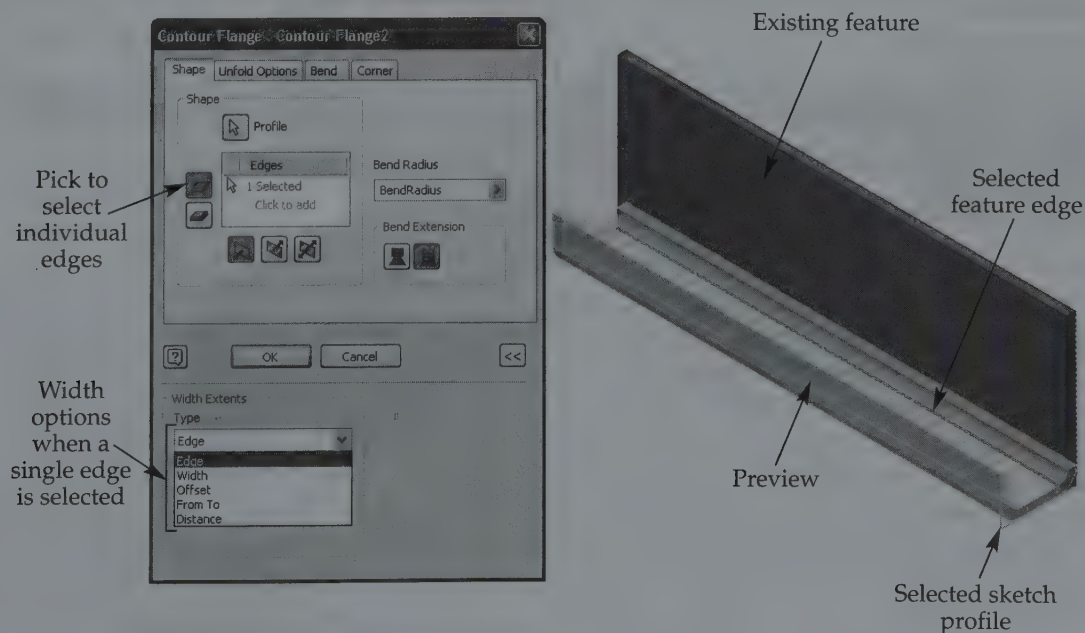
Figure 15-20.
Using the **Offset**
function of the **Width**
option to offset a
specific contour
flange width from a
selected point.
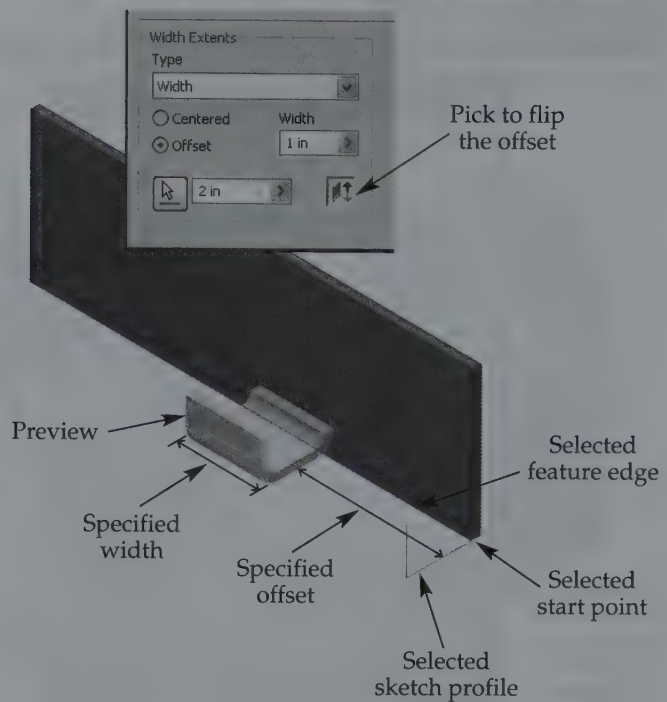
Select the **Offset** option to establish the contour flange width according to an offset from two points. See **Figure 15-21**. The first offset point corresponds to the location of the sketch profile on the selected edge. The second offset point is the opposite end of the edge. Use the **Offset1** and **Offset2** buttons to redefine the points if necessary. Specify offset values using the **OffSet1** and **OffSet2** text boxes.

Pick the **From To** option to establish the contour flange width between selected points, curves, faces, and/or planes. See **Figure 15-22**. Pick the **Offset1** button to select the first offset location, and then pick the **Offset2** button to select the second offset location.

Figure 15-21.
The **Offset** extent
option allows
the flange width
to change while
maintaining a
specified distance
between the flange
sides and the
selected geometry.
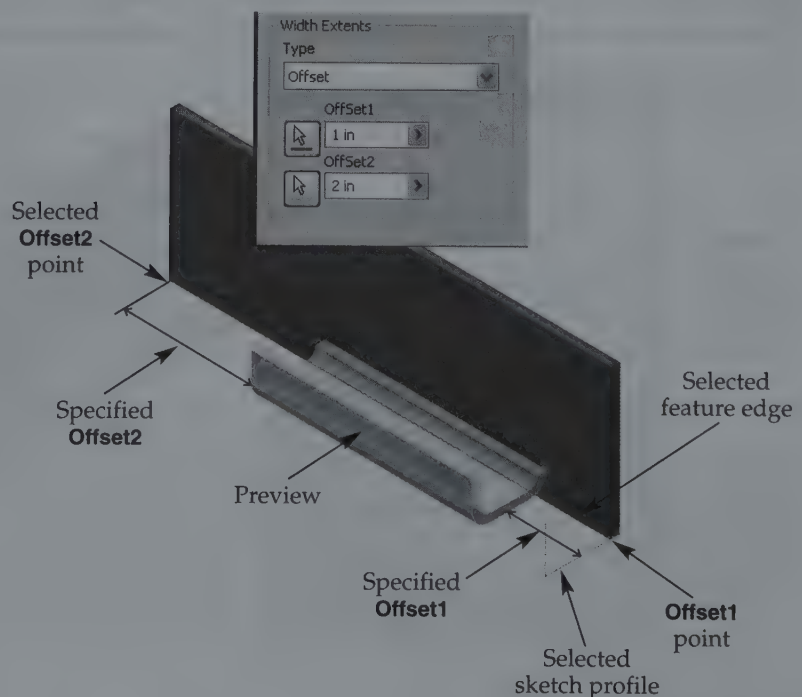
Figure 15-22.
The **From To** extent option establishes the flange width between two selected points, curves, faces, or as shown in this example, work planes.

Selected **Offset2**, a plane

Selected **Offset1**, a plane

Preview

Width Extents
Type
From To

Selected feature edge

Selected sketch profile
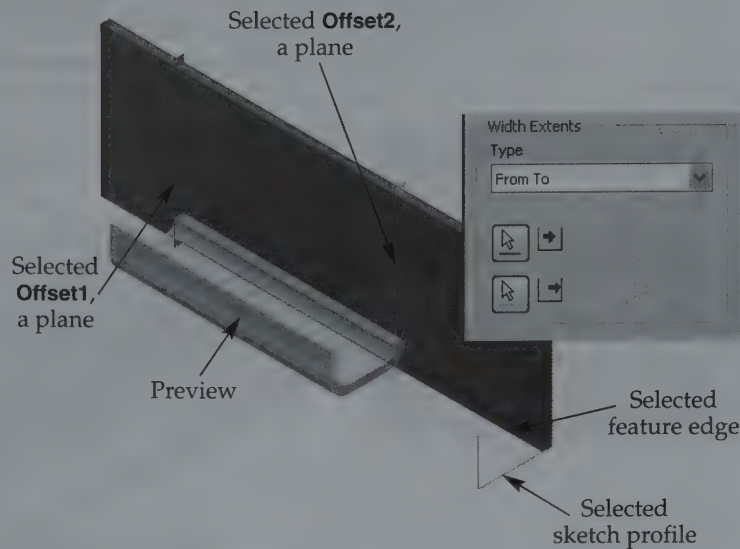
> **NOTE**
>
> You can use the **Distance** extent option with a selected edge, but the distance does not reference the edge like the other extent types.

Referencing Multiple Edges

To apply contour flanges along multiple edges, select the sketch profile and then use the **Edge Select Mode** button to pick individual edges, as shown in **Figure 15-23A**, or choose the **Loop Select Mode** button to select a closed loop. Options in the **Corner** tab, shown in **Figure 15-23B**, control what happens to intersecting bends. By default, the **Apply Auto-Mitering** check box is active, forming a miter at the intersection of bends, as shown in **Figure 15-23A**. Adjust the *miter gap* using the **Miter Gap** text box only if you want to override the default seam opening with a value greater than 0.

miter gap: Space between faces created during a corner seam or miter operation.

Deselect the **Apply Auto-Mitering** check box to remove mitered corners. Corners often fold together, creating self-intersecting bodies that cannot unfold into a flat pattern. Adjust individual corners when you see the preview, later in the design process, or when creating other features.

> **NOTE**
>
> Options in the **Width Extents** area are disabled when you pick multiple edges.

Editing Contour Flange Corners

During the preview of a contour flange, **Corner Edit** glyphs appear at the intersections of connected edges that form corners. Refer again to **Figure 15-23**. The **Apply Auto-Mitering** function, appropriate sheet metal rules, and if necessary, a miter gap override. This usually creates desirable corners. If not, pick a glyph to override corner parameters using the **Corner Edit** dialog box. See **Figure 15-24**. Use the **Overlap** check box, flyout, and **Gap** text box to modify the overlap and miter gap. The **Symmetric Gap** and **No Seam** overlap options are available when you edit a contour flange corner. Use the **Relief** check box, flyout, and **Relief Size** text box to modify the corner relief shape. Chapter 16 describes corner overlap options.

Figure 15-23.
A—Applying a contour flange to three selected edges. Bend intersections are mitered by default. B—Options in the **Corner** tab control how Inventor treats bend intersections. Notice the default values.

Pick to select individual edges

Pick to select a loop

When selecting individual edges, pick the edge perpendicular to the profile first

Corner edit glyph

Auto-mitered corners

A

Deselect to remove mitered corners

B
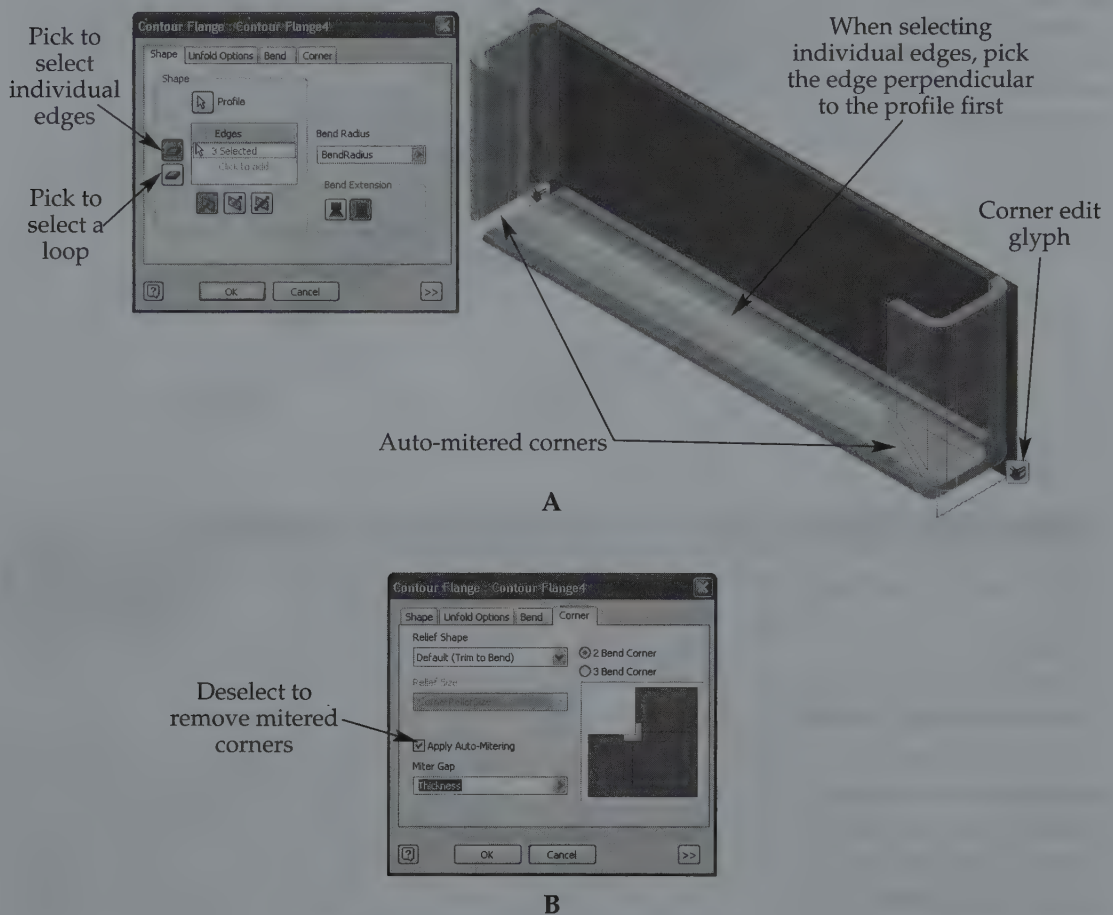
Figure 15-24.
Pick a **Corner Edit** glyph to display the **Corner Edit** dialog box, used to edit the characteristics of a specific contour flange corner.

Pick to enable the overlap flyout to modify the corner overlap

Pick to enable the relief flyout to modify the bend relief

Glyph indicates that the corner is edited

Finalizing Contour Flanges

The settings in the **Unfold Options**, **Bend**, and **Corner** tabs are the same as those located in the **Style and Standard Editor**. Adjust the settings only if you want to override the default sheet metal parameters. Pick the **OK** button to create contour flanges and exit, or pick the **Apply** button to create contour flanges and remain in the tool.

Exercise 15-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-6.

Flanges

Add a flange feature to produce a *flange* or similar sheet metal feature without developing a sketch. Access the **Flange** tool to create a flange using the **Flange** dialog box. See **Figure 15-25**. To add a flange to a single edge, use the **Edge Select Mode** button to pick the edge. Then pick the **More** button to specify the flange width. Flange width extents options are the same as those for creating a single contour flange.

To reference multiple edges, use the **Edge Select Mode** button to pick individual edges, or use the **Loop Select Mode** button to select a closed loop. See **Figure 15-26**. Flange corner formation and corner editing options are the same as those for referencing multiple edges to create contour flanges, except that additional overlap options are available for editing flange corners. Chapter 16 describes corner overlap options. Pick the **Flip Direction** button if flanges appear on the wrong side of the feature. Specify

flange: A feature added to reinforce or stiffen a part edge or to provide a surface for fastening or welding.

Ribbon
Sheet Metal > Create

Flange

Figure 15-25.
Using the **Flange** dialog box to create flanges.

Pick to select individual edges

Pick to select a closed loop

Pick to flip the side of the feature from which the flange forms

Bend edit gyph

Selected edge

Preview
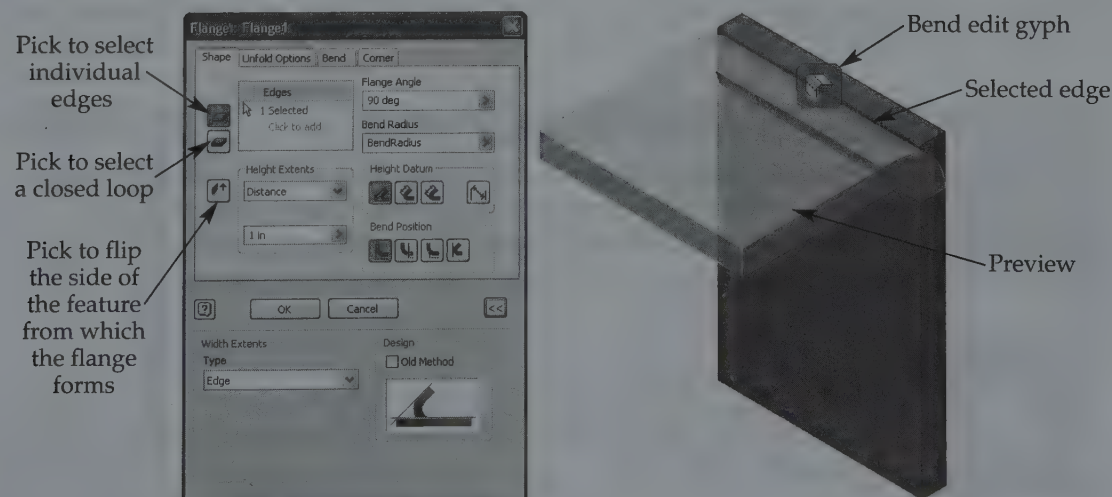
Figure 15-26.
A—An example of selecting multiple individual edges. B—Selecting a loop to add multiple flanges quickly.

Selected edges

A

Selected loop, top face

B
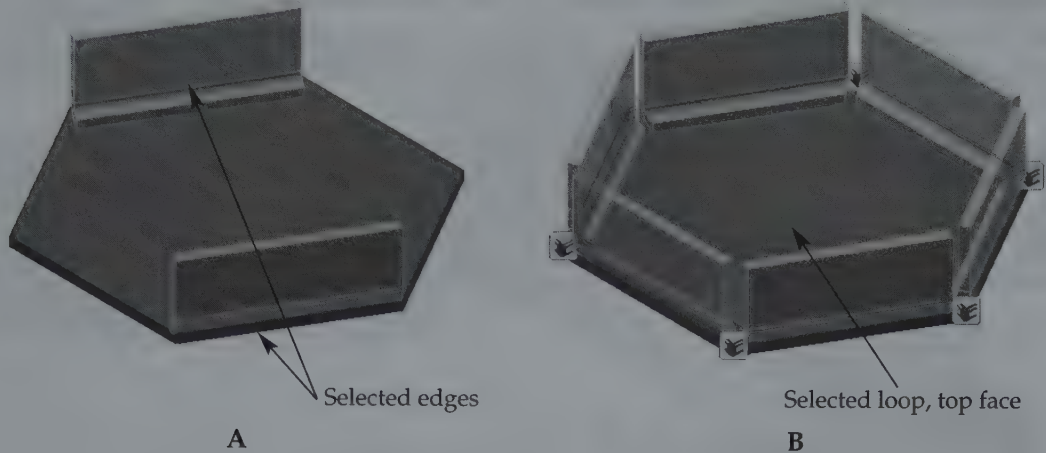
the flange angle using the **Angle** text box. Adjust the bend radius using the **Radius** text box only if you want to override the default bend radius, identified as BendRadius.

Flange Height

The **Height Extents** area controls the flange height, or projection from the edge. Use the default **Distance** option and specify the flange height from the selected edge using the **Distance** text box. Set the flange height *datum* by picking a button in the **Height Datum** area. **Figure 15-27** shows the effects of each height datum option. Next, define where the bend occurs by selecting a button in the **Bend Position** area. **Figure 15-28** shows the effects of each bend position option.

Select the **To** option from the **Height Extents** drop-down list to pick a point or vertex to control the flange height, without using a height datum. The select button becomes active, allowing you to pick a point or vertex. See **Figure 15-29A**. Specify a positive value in using the **Offset** text box to decrease the flange height from the selected point, as shown in **Figure 15-29B**, or specify a negative value to increase the flange height. The **Bend Position** options are available, but the **Height Datum** options are not.

datum: A theoretically exact point, axis, or plane from which the location or geometric characteristics of features originate.

Finalizing Flanges

The settings in the **Unfold Options**, **Bend**, and **Corner** tabs are the same as those located in the **Style and Standard Editor**. Adjust the settings only if you want to override the default sheet metal parameters. Pick the **Apply** button to create flanges and remain in the tool, or pick the **OK** button to create flanges and exit.

> **NOTE**
>
> The **Old Method** check box in the **Design** area of the expanded **Flange** dialog box is active when you open a file created using a version of Inventor prior to 2010. Select the **Old Method** check box to create flanges using the legacy method of design, which includes fewer options.

Exercise 15-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-7.

Figure 15-27.
Options for defining
the flange height
datum.

Height

Bend from the intersection of the
two outer faces — Aligned

Height

Bend from the intersection of the
two outer faces — Orthogonal

Height

Bend from the intersection of the
two outer faces — Orthogonal

Height

Bend from the intersection of the
two outer faces — Aligned

Height

Parallel to the flange detail
termination face
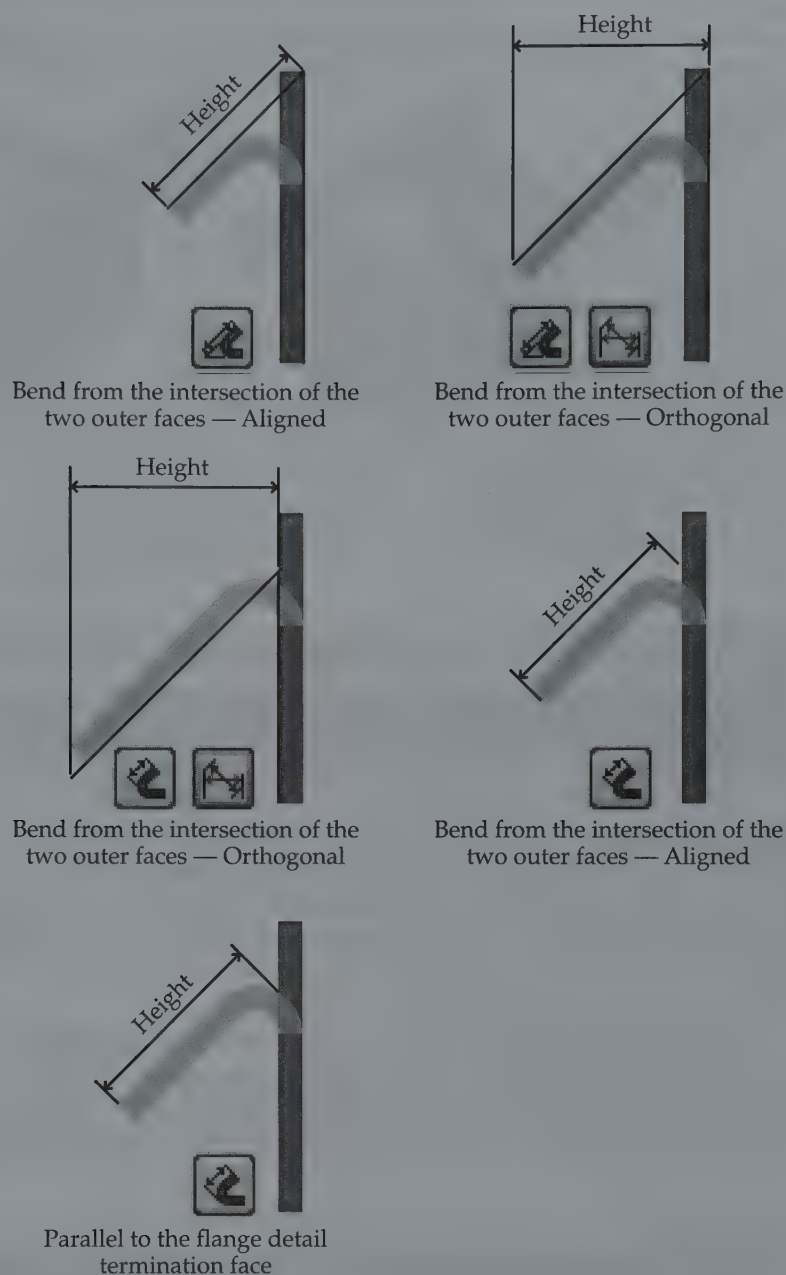
Figure 15-28.
Options for defining the flange bend position.

Inside of bend
face extents

Bend from the
adjacent face

Outside of bend
face extents

Bend tangent
to side face
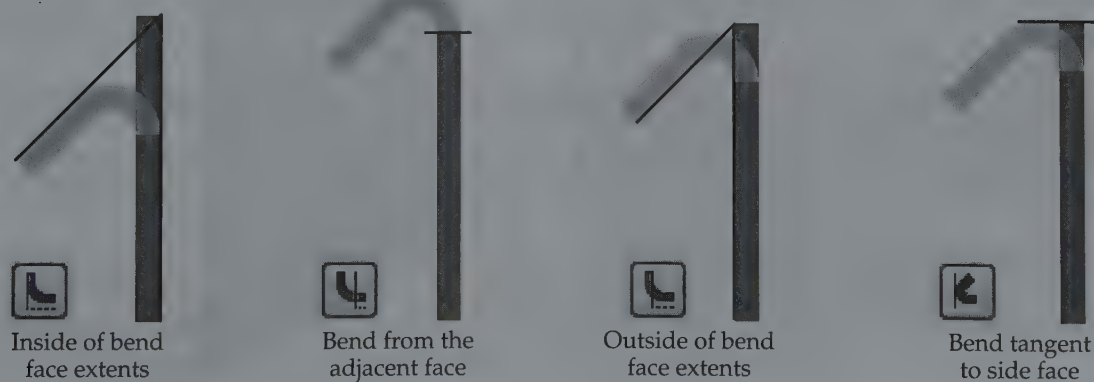
Figure 15-29.
A—Pick the **To** height extents option and select a point or vertex to define the flange height.
B—Specify an offset value to adjust the flange height from the vertex.

Selected point

Selected point

Offset

Height Extents
To
Offset
0 in

Height Extents
To
Offset
0.25 ir

A   B
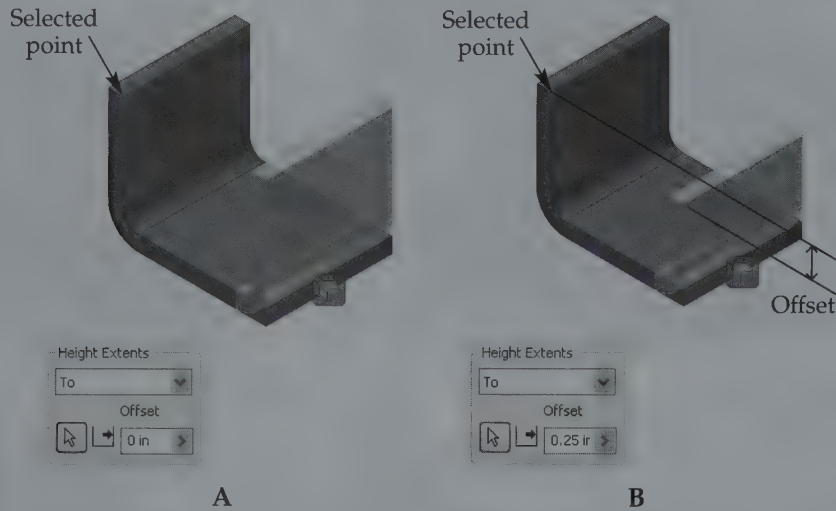
Hems

hem: Flanges used to add strength to or relieve the sharpness of exposed edges, or to connect separate edges or parts together.

Access the **Hem** tool to create a *hem* using the **Hem** dialog box. See **Figure 15-30**. Select a hem type from the **Type** drop-down list. **Figure 15-31** shows examples of hem shapes. Next, use the **Select Edge** button to pick an edge. The edge identifies the beginning of the hem width and the location of the bend. If necessary, pick the **Flip Direction** button to reverse the side of the feature on which the hem forms. Then pick the **More** button to specify the hem width. Hem width extents options are the same as those for creating a single contour flange or flange.

The **Gap** and **Length** text boxes are available for creating a single or double hem. The gap represents the opening between the inside hem face and the inside feature face. The length represents the amount of sheet metal material overlap used to create the hem. The **Radius** and **Angle** text boxes appear for creating a teardrop or a rolled

Figure 15-30.
Using the **Hem** dialog box to create hems. Notice the default gap and length values.

Select Edge button

Flip Direction button

Hem : Hem1

Shape | Unfold Options | Bend
Type
Single
Shape
Edge
Gap
Thickness * 0.50 ul
Length
Thickness * 4.0 ul

OK   Cancel   <<

Width Extents
Type
Edge

Selected edge

Preview
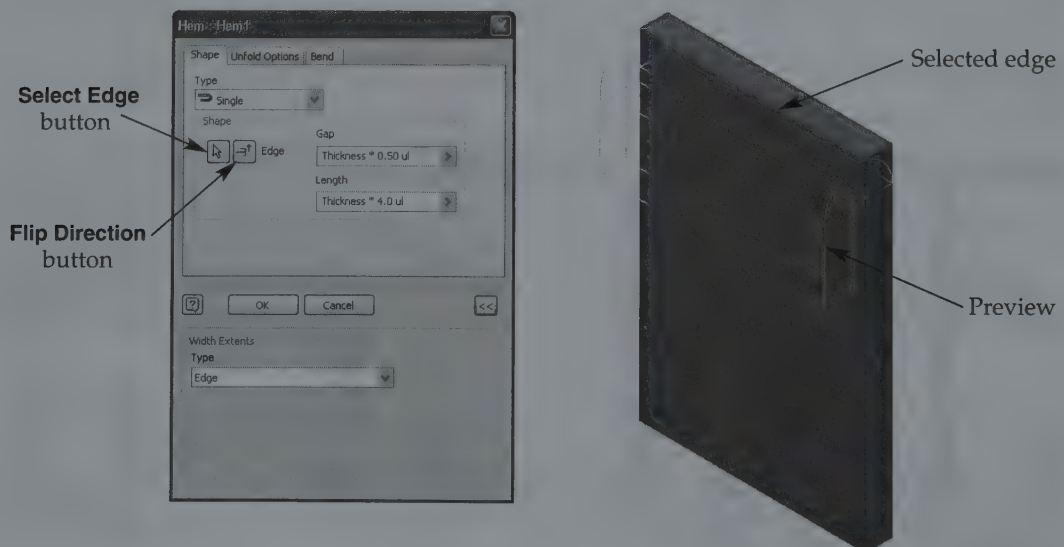
Figure 15-31.
Examples of common hems.

Single Hem   Teardrop Hem   Rolled Hem   Double Hem
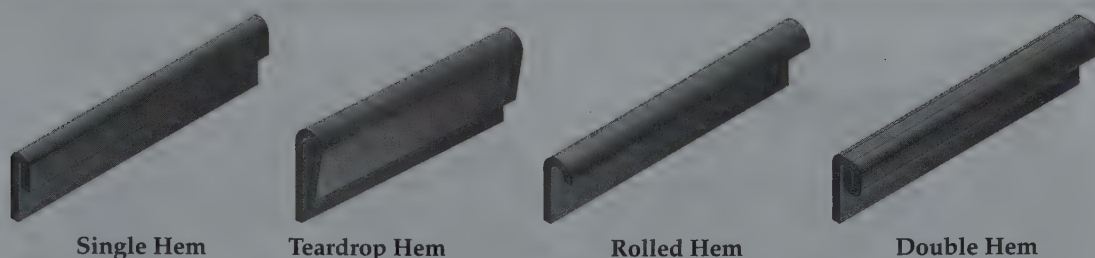
hem. The radius represents the bend radius. Adjust the radius only if you want to override the default bend radius, identified as BendRadius. The angle represents the amount of sheet metal material overlap used to create the hem and the angle at which the material is bent. The default angle for a teardrop hem is 190°. However, you can enter any value greater than 180° and less than 359°, depending on the thickness of the material and the size of the radius.

> **NOTE**
>
> Entering hem gap, length, radius, and angle values that are too small, too large, or otherwise inappropriate for the part will result in a hem that intersects existing features, usually causing a modeling error.

The settings in the **Unfold Options**, **Bend**, and **Corner** tabs are the same as those located in the **Style and Standard Editor**. Adjust the settings only if you want to override the default sheet metal parameters. Pick the **Apply** button to create contour flanges and remain in the tool, or pick the **OK** button to create contour flanges and exit.

Exercise 15-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-8.

Lofted Flanges

Inventor 2010
NEW

Use a *lofted flange* for applications such as a tapered column, reducer, transition piece, hopper, or any part that other sheet metal tools cannot build efficiently. Access the **Lofted Flange** tool to create a lofted flange using the **Lofted Flange** dialog box. See Figure 15-32. Use the **Profile 1** button to choose the first sketch profile. Then pick the second sketch profile using the **Profile 2** button.

lofted flange: A sheet metal feature that lofts one open or closed sketch profile to another on an offset plane or face to create sheet metal faces.

> **NOTE**
>
> Create a seam by including a gap in both sketch profiles, or use the **Rip** tool, described in Chapter 16, after forming the lofted flange.

Ribbon
Sheet Metal > Create

Lofted Flange

Use the thickness flip buttons to control the side of the sketch profile from which flange thickness occurs. Pick the **Both Sides** button to divide the thickness equally on both sides of the profile. Adjust the bend radius using the **Radius** text box only if you want to override the default bend radius, identified as BendRadius.

Press Break button

Multiple facets

36.000

24.000

Selected profile rectangle

Bend zone edit glyph

Selected profile (circle)

Press Brake Output

The **Press Brake** output button is active by default and forms multiple breaks, creating facets at each corner. Refer again to **Figure 15-32**. Select an option from the **Facet Control** drop-down list to establish the number of facets. The greater the number of facets, the smoother the corner transition between profiles and the greater the number of faces. Select the **Chord Tolerance** option to use a *chord tolerance*, the **Facet Angle** option to use a *facet angle*, or the **Facet Distance** option to use a *chord length*. Specify a reasonable value in the corresponding text box. A larger chord tolerance, facet angle, or facet distance creates fewer facets. Select the **Converge** check box to display relief in the folded model, in an attempt to form sharp corners in the flat pattern. See **Figure 15-33**.

Editing Press Brake Bends

During the preview of a contour flange, bend zone edit glyphs appear at loft transition corners. See **Figure 15-34A**. The lofted flange parameters and appropriate sheet metal usually create desirable facets. If not, pick a glyph to override the facet parameters using the **Bend Zone Edit** dialog box. Select the check box, and then define the desired facet control. In addition to the options available from the **Lofted Flange** dialog box, a **Number of Facets** method is available that allows you to specify the total number of facets using the text box. Individual facet bend edit glyphs also appear when you pick a **Bend Zone Edit** glyph. See **Figure 15-34B**. Select a bend edit glyph to override the corresponding bend parameters using the **Bend Edit** dialog box.

chord tolerance: The distance between an arc and facet edge at a line through the arc center point.

facet angle: The included angle with its vertex at the start of the facet, an arm tangent to an arc, and an arm collinear to the facet.

chord length: The linear distance between two points on a circle or arc.

Figure 15-33.
The effect of selecting or deselecting the **Converge** check box. Its main purpose is to form a sharper point at corners.

Converge Deselected

Converge Selected
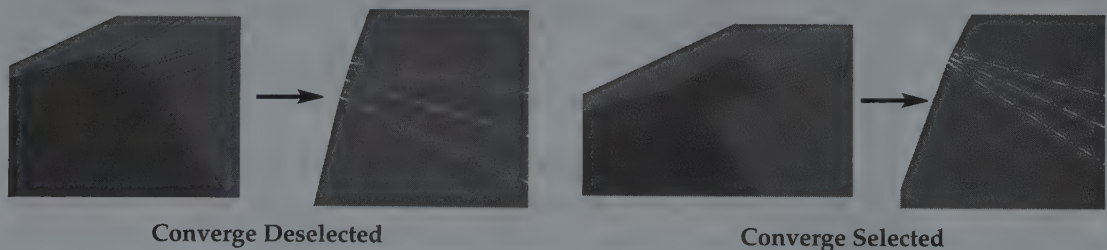
Figure 15-34.
A—Pick a bend zone edit glyph to display the **Bend Zone Edit** dialog box, used to edit facets.
B—Pick a bend edit glyph to display the **Bend Edit** dialog box, used to edit facet bends.

A

B
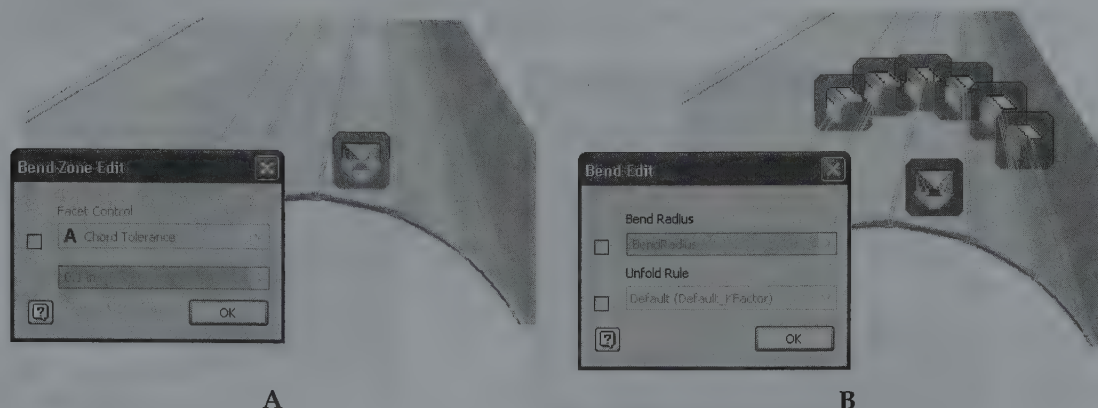
Figure 15-35.
An example of a
Press Brake and **Die
Formed** output.

Press Break   Die Formed
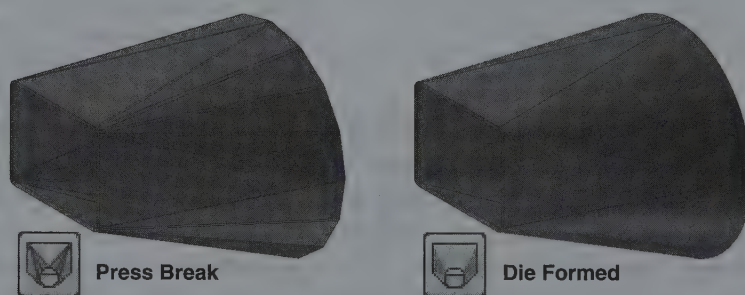
Die Formed Output and Finalizing

Select the **Die Formed** output to simulate forming the part using a die, forming block, or roller. See **Figure 15-35**. The **Die Formed** output creates a smooth transition between profiles and does not include facet options. Adjust the default settings in the **Unfold Options** tab only if you want to override the default unfold parameters. Pick the **OK** button to create the lofted flange and exit.

Exercise 15-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-9.

Contour Rolls

A *contour roll* is appropriate for turning applications such as rolled faces, beads, or similar features. Access the **Contour Roll** tool to create a contour roll using the **Contour Roll** dialog box. See **Figure 15-36**. If a sketch includes one profile and a line assigned the centerline format, the profile and axis are selected automatically. If a sketch includes one profile, but not a line assigned the centerline format, the **Axis** button is active, allowing you to pick the axis. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile to roll. You must then select the **Axis** button to pick the axis.

contour roll: A sheet metal feature that uses an open profile sketch rolled around an axis to create curved sheet metal faces.

Ribbon
**Sheet Metal
> Create**
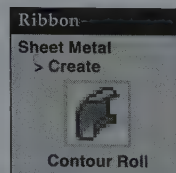
Contour Roll

Figure 15-36.
Using the **Contour Roll** dialog box to create a contour roll.

Open sketch profile selected
to roll (sketched on XY plane)

Centroid cylinder
reference

Thickness
flip
buttons

Direction
flip
buttons

Linear diameter
dimensional constraints
form by default when
you pick a centerline

Preview

Sketch includes a
projected axis (Y axis)
assigned a
centerline format
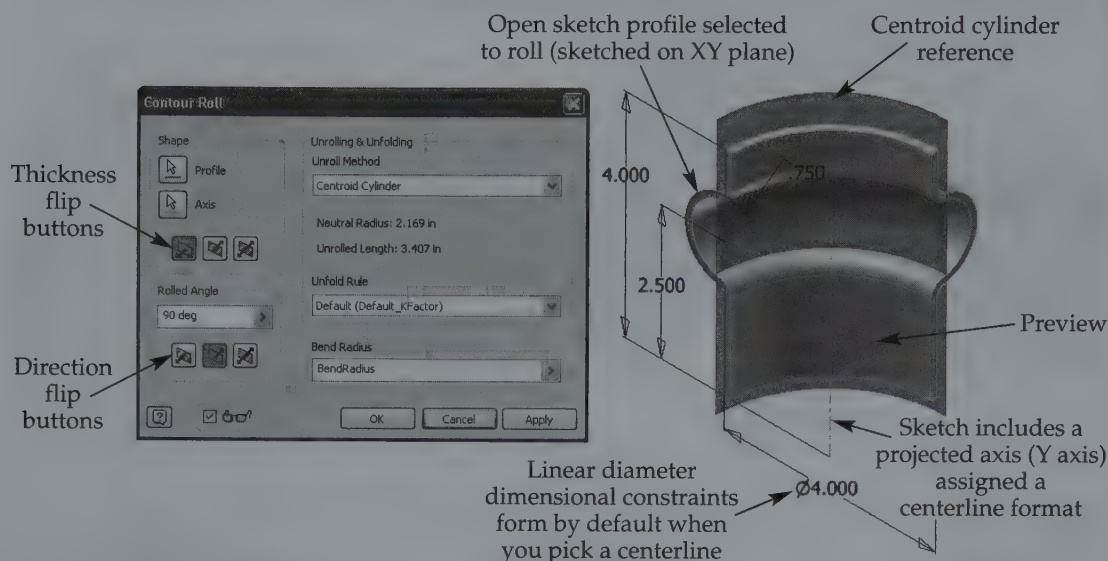
The axis defines the centerline of the roll. You must consider the location of the axis when developing the sketch. An axis can be any line in the sketch, an axis in the **Origin** folder of the browser, or a work axis. Inventor recognizes the sketch centerline or construction and centerline format as an axis, which can automate selection. Use the thickness flip buttons to control the side of the sketch profile from which face thickness occurs. Pick the **Both Sides** button to divide the thickness equally on both sides of the profile.

Use the **Roll Angle** text box to specify the angle, or amount, of roll with a value less than 360°. Use the direction flip buttons to control the side of the sketch plane from which roll angle occurs. Pick the **Mid-plane** button to divide the angle equally on both sides of the plane. The **Unroll Method** drop-down list allows you to specify additional unrolling, or unfolding parameters. The default **Centroid Cylinder** option calculates the *neutral radius* and *unfold length* based on your selections and the current settings. Pick the **Custom Cylinder** option to select a neutral axis, the **Developed Length** option to specify a *developed length*, or the **Neutral Radius** option to specify a neutral radius.

Adjust the bend radius using the **Radius** text box, and the unfold rule using the **Unfold Rule** drop-down list, only if you want to override the default bend radius and unfold parameters. Pick the **Apply** button to create the contour roll and remain in the tool, or pick the **OK** button to create the contour roll and exit.

neutral radius: The distance between the bend and the neutral axis.

unfold length (developed length): The length of the flat pattern required to form the roll.

Exercise 15-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 15-10.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. What are sheet metal parts?
2. Define flat pattern and give another name for it.
3. Define cutting in the context of sheet metal patterns.
4. Define bent.
5. What is a sheet metal rule?
6. What is the purpose of bend relief?
7. Where is the bend radius?
8. What is corner relief?
9. Define bend allowance.
10. Give the typical values for k-factor and briefly describe its function.
11. Where is the neutral axis?
12. What is a style library?
13. Describe a sheet metal face.
14. Where does a double bend occur?
15. What is a contour flange?
16. Briefly describe a miter gap.
17. Describe a flange.
18. What is a datum?
19. What is a hem?
20. Describe the basic function of a lofted flange.
21. What is a chord tolerance measurement?
22. Describe a facet angle.
23. Briefly describe a contour roll.
24. What is the neutral radius?
25. Describe unfold length and give another name for it.

Problems

1–4 Instructions: Follow the specific instructions for each problem.

1. Open the template file Sheet Metal-IN.ipt using the **Open** dialog box. Use the **Style and Standard Editor** to create and save a sheet metal rule named 26 GAGE STEEL. Select the Steel material and enter a thickness of .0179. Leave all other values set as default. Save and close the template file.

2. Open the template file Sheet Metal-IN.ipt using the **Open** dialog box. Use the **Style and Standard Editor** to create and save a sheet metal rule named 4 GAGE 6061. Select the Aluminum-6061 material and enter a thickness of .2043. Leave all other values set as default. Save and close the template file.

3. Open the template file Sheet Metal-mm.ipt using the **Open** dialog box. Use the **Style and Standard Editor** to create and save a sheet metal rule named 0.5 430. Select the Stainless Steel material and enter a thickness of 0.5. Leave all other values set as default. Save and close the template file.

4. Open file P4-10.ipt and save it as P15-4.ipt. In the P15-4.ipt file, convert the model from a modeling application to a sheet metal application. Create a contour flange offset toward the inside of the sketch .5″ midplane as shown. Resave the part.

5–6 Instructions:

- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*

- Add the dimensional constraints shown.
- Add as much information as possible to the *iProperties* dialog box. Assign the specified material and color to the part.
- Follow the specific instructions for each problem to create the features.

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

5. **Title:** CLIP
   Units: Inch
   Template: Sheet Metal-IN.ipt
   Part Number: IAA-036-01
   Project: CLIP
   Material: According to sheet metal rule
   Color: As Material
   Sheet Metal Rule: 11 GAGE STEEL
   Save as: P15-5.ipt
   Specific Instructions: Create the sketch shown on the XZ plane. Use the **Face** tool to create a face offset above the sketch plane. Create a contour flange offset toward the inside of the sketch, along the selected edge, using the **Offset** width extents type and the information shown. The final part should look like the part shown.

3.000

3.000

Projected center point

Endpoint coincident to midpoint

Sketch plane

1.500

Perpendicular

1.500

Perpendicular

Offset2, .5" offset

Selected edge

Offset1, .5" offset
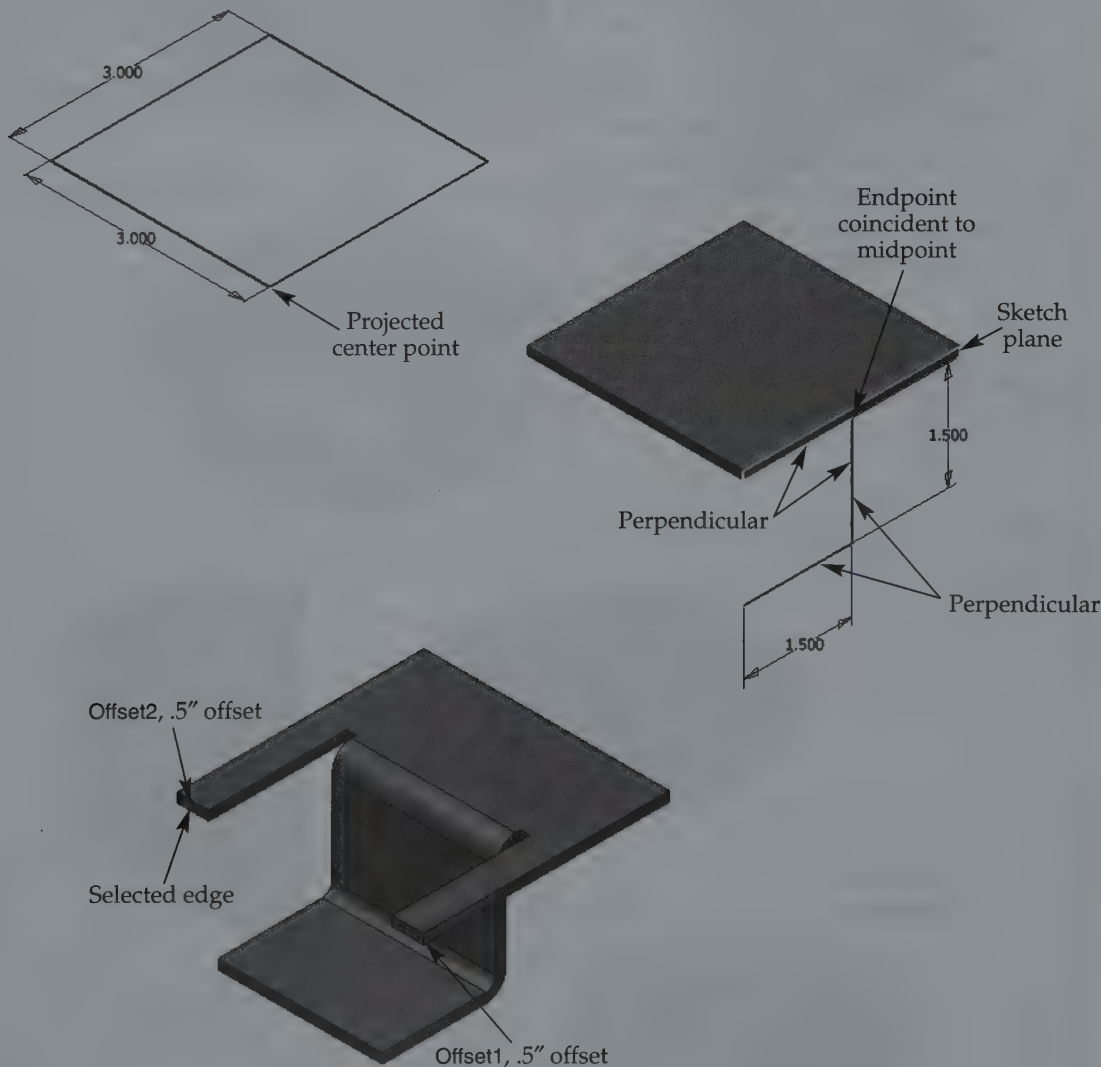
6. **Titles:** INTERIOR HUB FLANGE and EXTERIOR HUB FLANGE
   Units: Inch
   Template: Sheet Metal-IN.ipt
   Part Numbers: IAA-037-01 (interior) and IAA-037-02 (exterior)
   Project: GENERATOR
   Material: According to sheet metal rule
   Color: As Material
   Sheet Metal Rule: 11 GAGE STEEL
   Save as: P15-6INT.ipt (interior) and P15-6EXT.ipt (exterior)
   Specific Instructions:
   A. Create the sketch of the octagon (use the **Polygon** tool) on the XZ plane, as shown in A.
   B. Use the **Face** tool to create a face offset above the sketch plane, as shown in B.
   C. Create the sketch shown on the XY plane, as shown in B.
   D. Create a contour flange offset toward the outside of the sketch, along the selected loop, using the information shown in C.
   E. Open a sketch on the face shown and sketch the circle shown in D.
   F. Cut the circle through the thickness of the material.
   G. Use the **Circular Pattern** tool to pattern the cut as shown in E. Select the default Y axis as the rotation axis, specify a count of 8, a fitted position method, and 360° angle.
   H. Cut a Ø1.5″ circle, with a center coincident to the projected center point as shown in E. The final part should look like the part shown.
   I. Save the part as P15-6INT.ipt.
   J. Edit the contour flange sketch as shown in F. The final part should look like the part shown in G.
   K. Save the part as P15-6EXT.ipt.
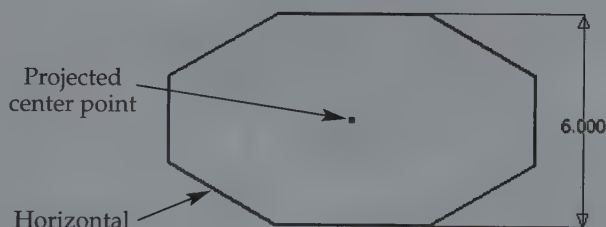
A
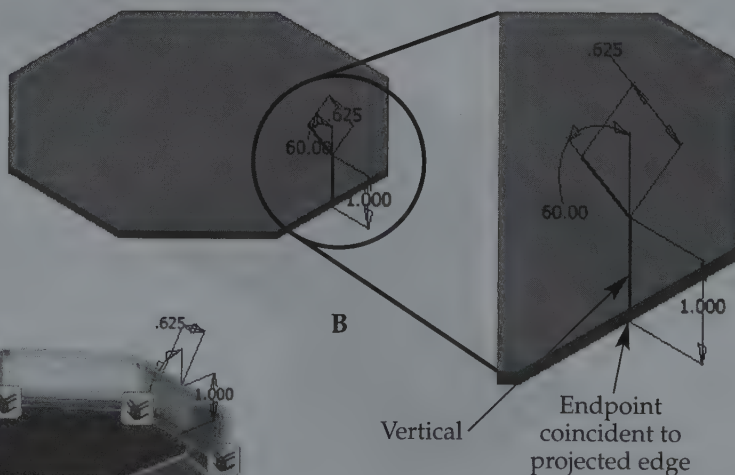
B
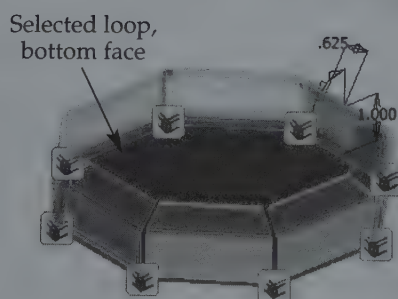
C

(Continued)

Selected loop,
bottom face

.625

2.000

Projected
center point

D

E

.750

60.00

1.500

.750

60.00

1.500
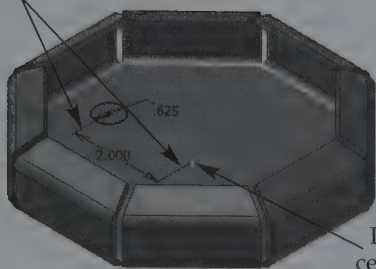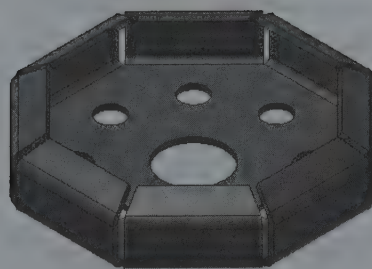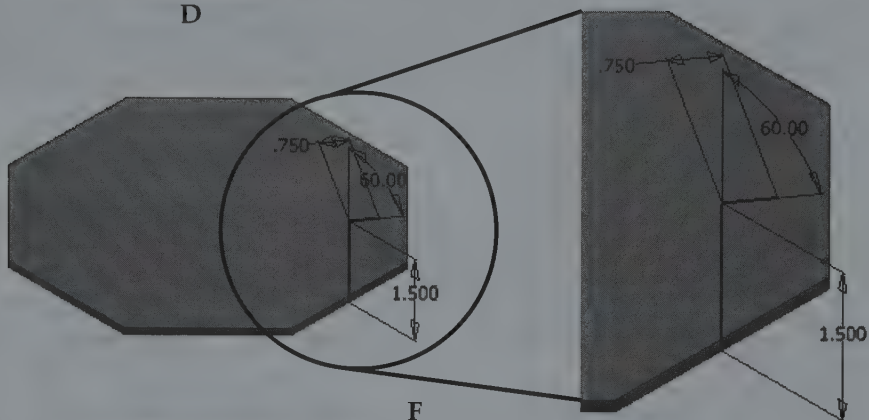
F
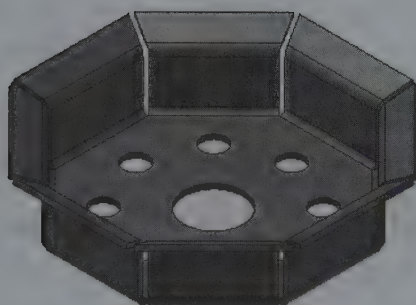
G

7. **Title:** FRAME BRACKET
   Units: Inch or metric
   Template: Sheet Metal-IN.ipt or Sheet Metal-mm.ipt
   Part Numbers: IAA-567-01
   Project: CART
   Material: According to sheet metal rule
   Color: As Material
   Sheet Metal Rule: Create an appropriate sheet metal rule for 18-gauge cold-rolled steel
   Save as: P15-6.ipt
   Specific Instructions: Create a sheet metal part according to the specifications above and the drawing shown.

2X Ø1.015 MIN [25.8]
2X R.750 [19.1]
2X Ø.280 WELD HOLE [7.1]
2X R.750 [19.1]
DOWN 90° R.525 MIN [DOWN 90° R13.3 MIN]
6.916 [175.7]
Ø.188 [4.8]
4.058 [103.1]
2X 4.916 [124.9]
3.458 [87.8]
2.029 [51.5]
DOWN 90° R.525 MIN [DOWN 90° R13.3 MIN]
2X 2.458 [62.4]
8.416 [213.8]
4.208 [106.9]
[BEND 2X R.525 MIN] [13.3]
.219 [5.6]
7.937 [201.6]
10.000 [254]
(4.250) [108]

NOTES:
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
3. DIMENSIONS IN [ ] ARE MILLIMETERS.

UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES (IN.)
TOLERANCES:
X ±.1
.XX ±.01
.XXX ±.005
ANGULAR ±10'
FINISH: 62μin.

THIRD ANGLE PROJECTION

APPROVALS		DATE
DRAWN	DPM	7/22/2009
CHECKED	JAJ	7/22/2009
APPROVED	JAJ	7/22/2009

MATERIAL: 18 GAGE CRS
FINISH: ALL OVER

G-W Goodheart-Willcox
TITLE: FRAME BRACKET

SIZE B
CAGE CODE
DWG NO IAA-567-01
REV 0
SCALE 1:2
SHEET 1 OF 1
DO NOT SCALE DRAWING

Additional Sheet Metal Tools

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Add folds and bends.
✓ Create cuts and use the **PunchTool**.
✓ Place corner rounds and chamfers.
✓ Use the **Corner Seam** and **Rip** tools.
✓ Unfold and refold for feature development.
✓ Create and work with flat patterns.

This chapter describes additional common sheet metal features and tools sometimes required to correct modeling errors. You will also learn to unfold and refold formed sheet metal products during part development to create features in reference to the flat pattern. This chapter ends by describing flat pattern tools and techniques.

Folds

To create a fold, first sketch a single line that extends across the entire portion of the feature to fold, without extending past the sketch plane. Then access the **Fold** tool to fold the part using the **Fold** dialog box. See **Figure 16-1**. Use the **Bend Line** button to select the sketch line where the fold will occur. The **Flip Side** and **Flip Direction** buttons control where the angle of the fold applies.

Ribbon
Sheet Metal > Create

Fold

Pick the **Center Line of Bend** button to use the bend line as a centerline, bending the material equally on each side of the bend line. Select the **Start of Bend** button to start the bend at the bend line, placing the bend on the portion of the material that is bent. Pick the **End of Bend** button to begin the bend on the opposite side of the bend line, placing the bend on the portion of the material that is unbent.

Use the **Fold Angle** text box to specify a fold angle, or included bend angle. **Figure 16-2** shows some of the bends you can create using a 90° bend angle. Adjust the bend radius using the **Bend Radius** text box only if you want to override the default bend radius. Pick the **Apply** button to create the fold and remain in the tool, or pick the **OK** button to create the fold and exit.
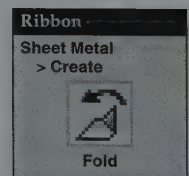
Figure 16-1.
Using the **Fold** dialog box to create a fold at a sketch line. Use the direction and side icons to help determine the correct fold from the bend line.

Flip Side button

Flip Direction button

Centerline of Bend button

Start of Bend button

End of Bend button

Bend Line

Flip Controls

Fold Location

Fold Angle — 90.0

Bend Radius — BendRadius

Selected sketch line

.500

Fold location extents (centerline of bend active)

.375

Side   Direction
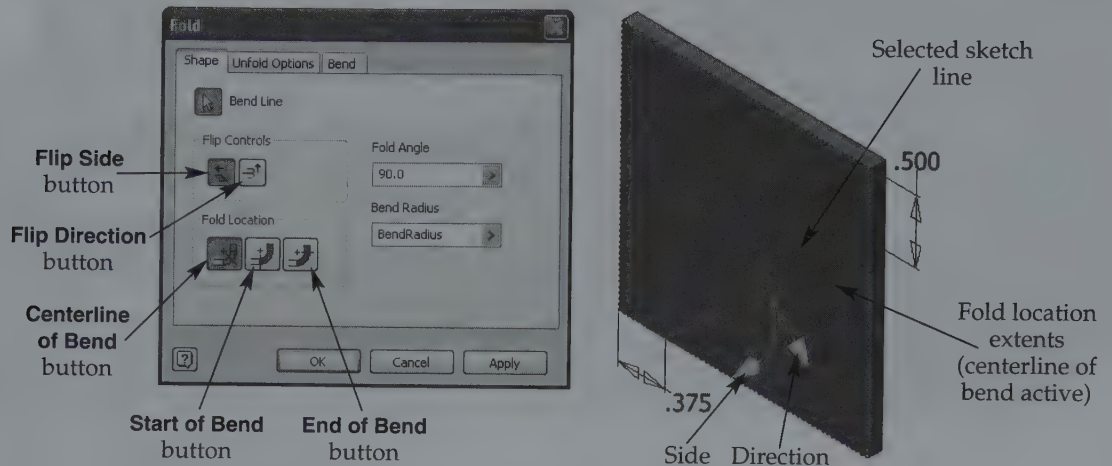
Figure 16-2.
Examples of 90° folds created by adjusting fold parameters. You can use any fold angle appropriate for the sheet metal parameters.
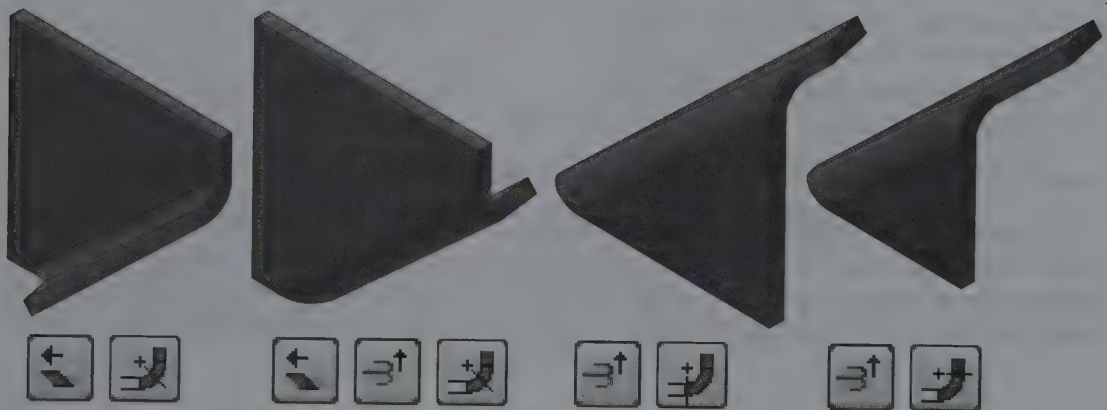
Exercise 16-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-1.

Bends

A bend is usually created during part development using sheet metal feature tools. However, it may be necessary to add a bend when a bend does not form, when you are building a sheet metal part from a converted non-sheet metal part, or to place a double bend. See **Figure 16-3**. Access the **Bend** tool to add bends using the **Bend** dialog box. See **Figure 16-4**. Use the **Select Edges** button to pick a single edge, as shown in **Figure 16-3A**, or two edges, as shown in **Figure 16-3B**.

Options for placing a double bend using the **Bend** tool are the same as those for creating a double bend with the **Face** tool. The settings in the **Radius** text box, and
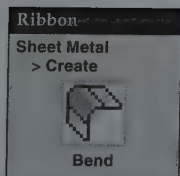
Figure 16-3.

Figure 16-3.
A—Bending a sharp corner so that the part can unfold into a flat pattern. B—Adding a 90°
double bend between two parallel faces.

Selected edge

Bend

Original Part

Complete Part

A

Selected edges

90 Degree bend

Original Part

Complete Part

B
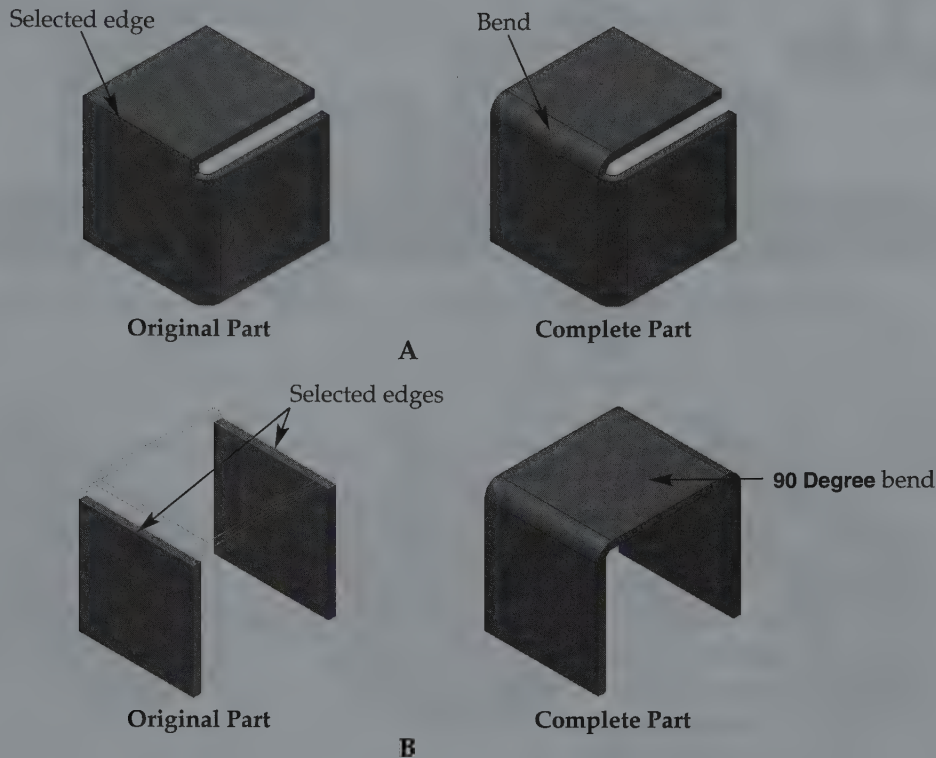
Figure 16-4.
The **Shape** tab of the
Bend dialog box.
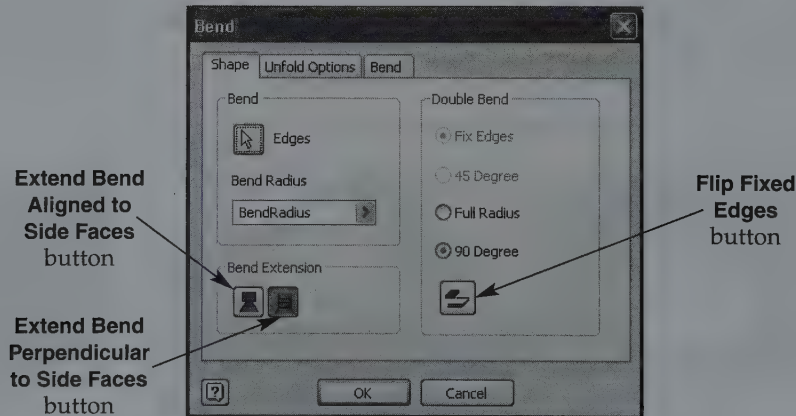
Extend Bend
Aligned to
Side Faces
button

Extend Bend
Perpendicular
to Side Faces
button

Flip Fixed
Edges
button

Unfold Options and **Bend** tabs are the same as those located in the **Style and Standard Editor**. Adjust the settings only if you want to override the default sheet metal parameters. Pick the **OK** button to create the fold and exit, or pick the **Apply** button to create the fold and remain in the tool. Pick the **OK** button to exit.

NOTE

To change a bend to a corner seam, right-click on the bend in the browser and select **Change to Corner**. The bend is removed and the **Corner Seam** dialog box appears.

Cuts

A sheet metal cut uses a closed profile sketch to remove material from existing features. Access the **Cut** tool to create a cut using the **Cut** dialog box. See **Figure 16-5**. If a sketch includes one profile, the profile is selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick profile(s). The **Distance**, **To Next**, **To**, **From To**, and **All** extents options behave the same as when adding a cut extrusion using the **Extrude** tool.

Figure 16-5 shows an example of using the **Distance** extents option with the default Thickness value to cut through the material thickness, even after you make changes to the thickness parameter. **Figure 16-6** shows an example of another application and examples of the other extents options. When the **Distance** extents option is selected, the **Flip** and **Both Sides** buttons are available to adjust where the cut occurs. When the **To** or **From To** extents option is selected, and the selected face does not intersect the cut path, pick the **Terminate feature on extended face** check box.

Pick the **Cut Across Bend** check box, located in the **Shape** area, to cut through the material thickness and wrap the cut around bends and faces that are not coplanar to the sketch plane. See **Figure 16-7**. You can override the default distance to cut past features if necessary using the text box in the **Extents** area. Pick the **OK** button to create the cut.

Figure 16-5.
Using the **Cut** dialog box to cut a sketch profile through sheet metal features.

Selected sketch profile

Preview
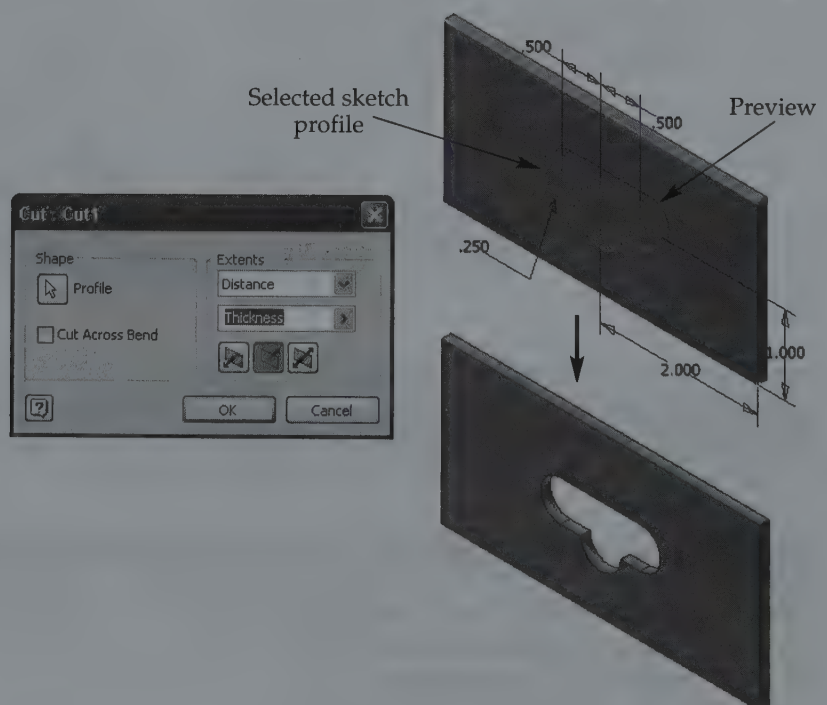
Figure 16-6.
Examples of cutting using each extents option.

Specified distance

Distance

Next possible face or plane

To Next

To plane

From plane

To plane (rear)

To

From To

All
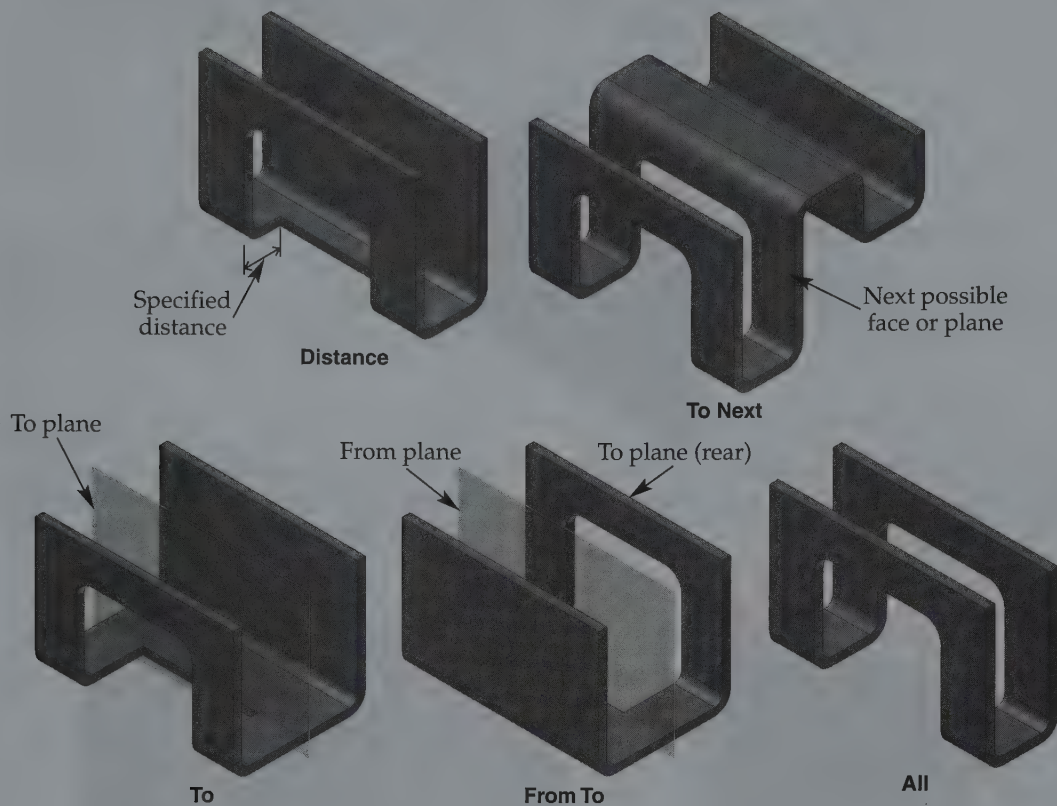
Figure 16-7.
Using the **Cut Across Bend** option to cut a circle through and along bends and faces that are not coplanar to the sketch plane. Notice that the flat pattern retains the correct circle shape.

1.500

1.500

1.000

Cut through thickness

Cut wraps around faces

Cut Sketch

Cut
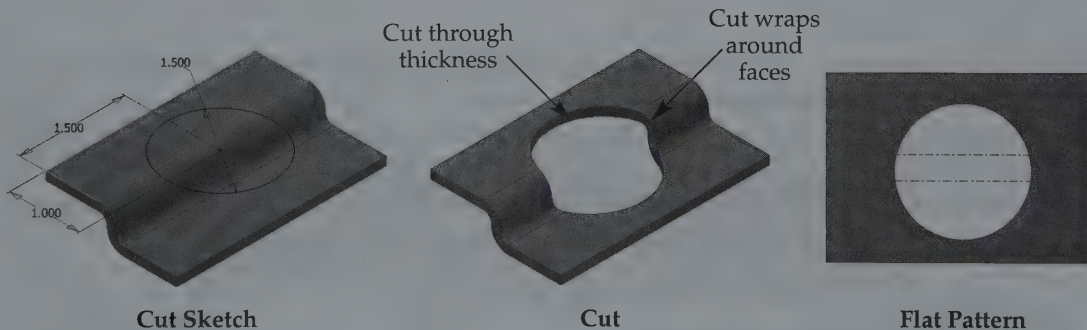
Flat Pattern

Exercise 16-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-3.

Projecting Flat Pattern Geometry

Sometimes when developing a sheet metal sketch, it is necessary to reference flat pattern geometry without creating a flat pattern or unfolding the part. While sketching, access the **Project Flat Pattern** tool and then pick surfaces to project the corresponding flat pattern geometry onto the sketch plane at the current design state. Selecting a

Ribbon

**Sketch
> Draw**

Project Flat Pattern

Figure 16-8.
A—Projecting flat pattern geometry onto a sketch plane. Notice that bend centerlines also project. B—Dimensioning to flat pattern edges. C—Using the **Cut** tool to cut across bends. Notice that the specified .5″ sketch dimension accurately wraps to the vertical face.

A

B

C
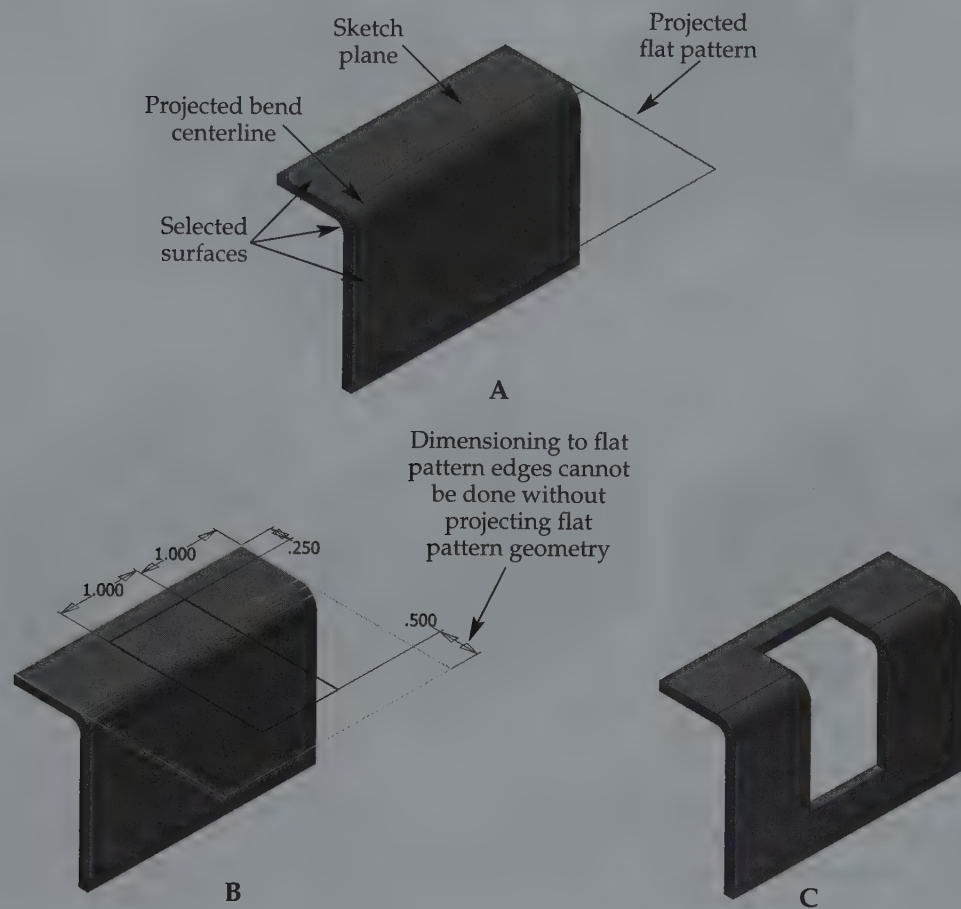
single face sometimes projects other tangent faces, though often you must pick each surface individually. See **Figure 16-8A**.

You can use projected flat pattern geometry for a variety of applications. One of the most common applications is dimensioning a sketch profile to flat pattern edges. See **Figure 16-8B**. When used with the **Cut** tool and **Bend Across Cut** option, for example, the cut occurs according to flat pattern requirements. See **Figure 16-8C**.

Exercise 16-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-4.

Using the PunchTool

sheet metal punch:
A press or similar tool used to form a specific shape or hole in sheet metal.

The **PunchTool** uses sheet metal iFeatures to replicate the process of using a *sheet metal punch* tool. Using the **PunchTool** is similar to inserting an iFeature. However, **PunchTool** iFeatures reference sketched points for positioning.

Creating PunchTool iFeatures

You can create and store a **PunchTool** iFeature using techniques similar to creating and storing standard iFeatures, including table-driven iFeatures. However, **PunchTool** iFeatures have specific additional requirements and options. **PunchTool** iFeature sketch geometry must include a point created using the **Point, Center Point** tool in order for you to position the iFeature using the **PunchTool**. See **Figure 16-9A**. Sheet metal features, such as the cut shown in **Figure 16-9B**, complete the iFeature geometry.

Access the **Extract iFeature** tool to extract an iFeature using the **Extract iFeature** dialog box. See **Figure 16-10**. To extract a **PunchTool** iFeature, pick the **Sheet Metal Punch iFeature** radio button. Then select features to extract. Set size parameters and position geometry using the same methods as when extracting a standard iFeature. If necessary, enter a *punch ID* in the **Specify Punch ID** text box. Drawing tools are available that allow

---

Figure 16-9.
A—An example of the base feature sketch used to develop a **PunchTool** iFeature. The sketch point positions the iFeature when punched using the **PunchTool**. B—The completed feature created using the **Cut** tool, ready for iFeature extraction.

4.000
1.000
3.000
.500
2.000
Sketch center point
A

Face not included in iFeature extraction
iFeature
B
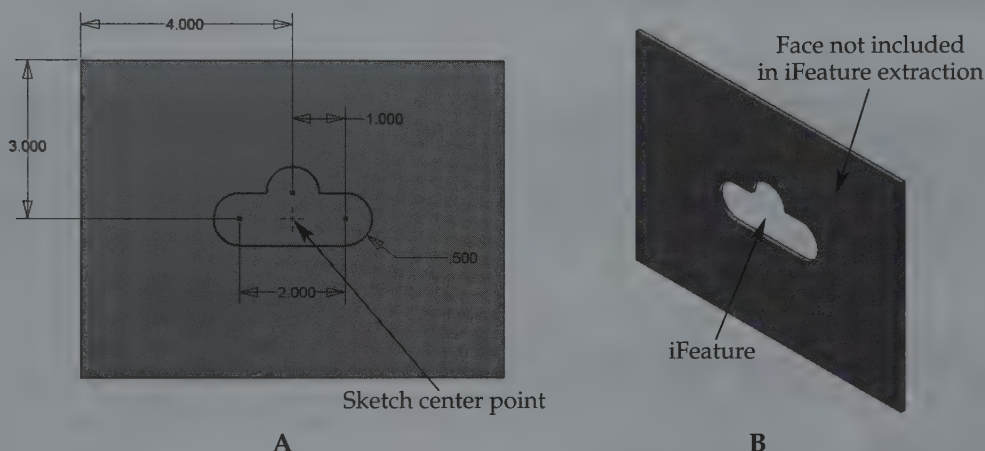
Figure 16-10.
The **Extract iFeature** dialog box with the **Sheet Metal Punch iFeature** radio button selected.

Enter a punch identification value

Enter a default punch depth

Pick to select a sketch from the browser
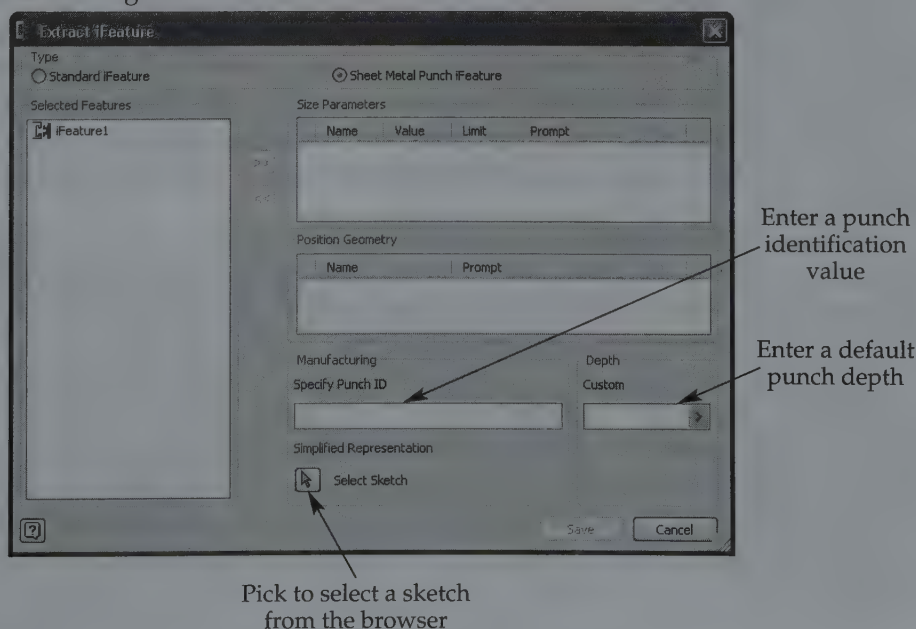
you to extract the specified punch ID into a specific note or for use in a form of tabular dimensioning.

The settings in the **Style and Standard Editor** control how sheet metal punches created using the **PunchTool** appear in the flat pattern. If you select the **2D Sketch Representation** or **2D Sketch Rep and Center Mark** option in the **Flat Pattern Punch Representation** area of the **Sheet** tab, a sketch or a sketch and a center mark appear in place of the punched feature in the flat pattern. To specify which sketch appears, pick the **Select Sketch** button and select a sketch from the browser. If you do not select a sketch, no sketch appears in the flat pattern.

If appropriate, enter the default custom punch depth in the **Custom** text box. You can adjust the depth when you insert the iFeature using the **PunchTool**. The depth does not change the appearance of the iFeature. A depth is typically appropriate when you are preparing a 2D drawing to identify the punch depth. You can use drawing tools to extract the specified punch depth as needed.

NOTE

Table-driven iFeature tables include Sheet Metal Rule, Sheet Metal Unfolding Rule, and Flat Pattern Orientation columns.

PROFESSIONAL TIP

Save all **PunchTool** iFeatures in a specific location. Supplied **PunchTool** iFeatures are located in the following path: Program Files/Autodesk/ Inventor 2010/Catalog/Punches.

Exercise 16-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-5.

Punching

To prepare a feature for punching using an sheet metal punch iFeature, sketch points using the **Point, Center Point** tool. See **Figure 16-11**. You can include multiple points in a sketch to punch the same iFeature multiple times in a single operation. Access the **PunchTool** tool to display the **PunchTool Directory** dialog box. Use the dialog box to locate and open an iFeature punch file anywhere on your computer or the network.

Once you select an iFeature, the **PunchTool** dialog box and a punch preview appear at each unconsumed sketch point. The **Preview** tab, shown in **Figure 16-12A**, displays the contents of the folder in which the selected punch resides, and a preview of the selected punch. Select a different available punch if appropriate.

The **Geometry** tab, shown in **Figure 16-12B**, allows you to pick additional sketch points using the **Select Centers** button. To disable punching at certain center points, pick the **Select Centers** button, hold down [Ctrl], and pick the center points to deselect. Use the **Angle** text box to specify the punch angle. The **Geometry** tab also helps satisfy *dangling geometry*, such as faces, edges, angles, or other reference points. To satisfy dangling geometry, pick a name from the **Dangling Geometry** list box and follow the prompt.

dangling geometry: A condition that results when additional positioning information is required in order for iFeature insertion to occur; primarily due to issues with the initial iFeature sketch and existing feature geometry.

Figure 16-11.
An example of a feature with a sketch, ready to receive a punch.

4.000

Point, Center Point

3.000
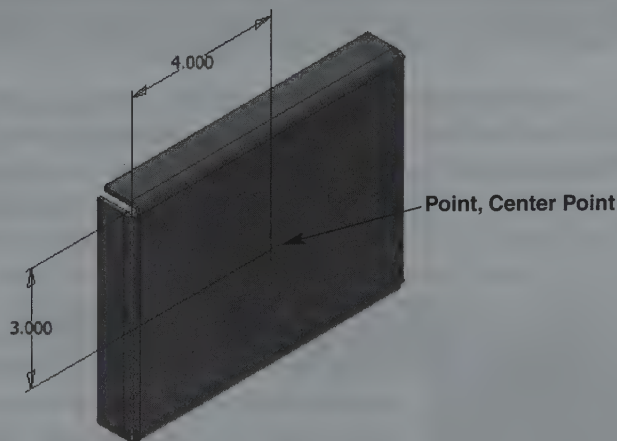
Figure 16-12.
A—The **Preview** tab of the **PunchTool** dialog box contains a list of available punches and a preview of the selected punch. B—The **Geometry** tab allows you to add one or more punches to the selected part and define the punch angle. C—The **Size** tab allows you to change the size of included parameters.

iFeatures found in the folder of the selected iFeature

Included iFeature parameters

Preview window

Selectable when linked or embedded file is available that describes placement techniques

A   B   C
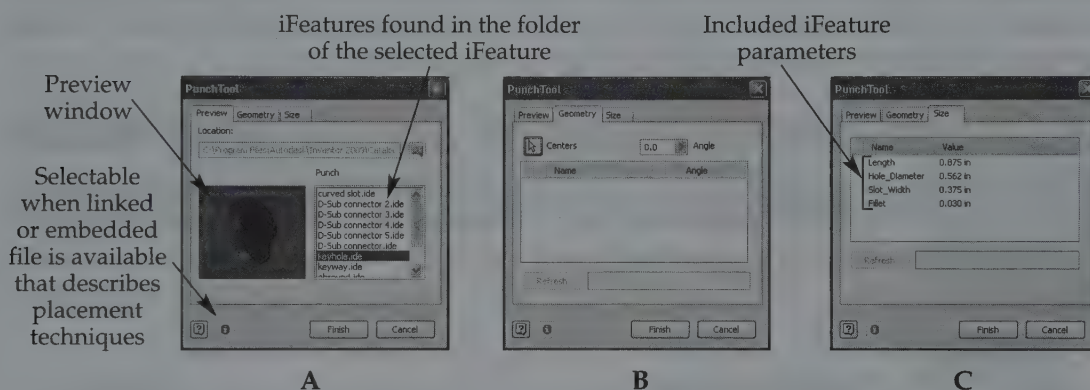
The **Size** tab, shown in **Figure 16-12C**, lists size parameters specified during the iFeature definition in the **Size** list box. The prompts guide you through the iFeature sizing process. If you specified the **None** size limit option, enter any appropriate value. If you specified the **Range** size limit option, enter a value within the range. If the specified size is outside the range, the value is red. If you specified the **List** size limit option, choose a value from the drop-down list. A punch can also be table-driven. Pick the **Refresh** button whenever you reposition or resize the punch to preview the changes. Pick the **Finish** button to complete the punch operation.

Exercise 16-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-6.

corner round: A curve placed at an inside (fillet) or outside (round) sheet metal corner.

Ribbon

Sheet Metal > Modify
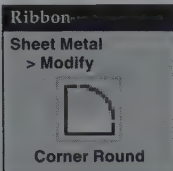
Corner Round

A *corner round* is common for removing sharp points and edges, or for applying a required radius. Access the **Corner Round** tool to place corner rounds using the **Corner Round** dialog box. See **Figure 16-13**. Use the **Corner Round** dialog box to create fillets and rounds with various radii without continually re-accessing the **Corner Round** tool. You can add all of the fillets and rounds shown in **Figure 16-13**, for example, using a single corner round operation, which displays as a single feature in the browser.

PROFESSIONAL TIP

If you plan to suppress certain corner rounds or want to have additional corner round features access the **Corner Round** tool more than once.

Specify a radius using the **Radius** column text box. Then use the **Corner** radio button to select individual corners, or pick the **Feature** radio button to assign the same radius to all corners of a selected feature. To apply fillets and rounds with different radii using a single **Corner Round** operation, pick the **Click to add a corner set** button. Then enter a radius and select corners or features. Continue adding fillets and rounds as required. Pick the **OK** button to create the feature and exit.

NOTE

Deselect chosen corners by holding down [Ctrl] and picking the corners. To delete a corner listed in the **Corner** list box, pick the corner and press the [Delete] key.

Exercise 16-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-7.

Figure 16-13.
Using the **Corner Round** dialog box to apply several corner rounds with different radii. Notice that you can select only feature corners.
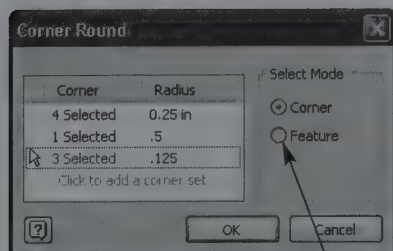
Pick to assign a radius to all corners of a selected feature
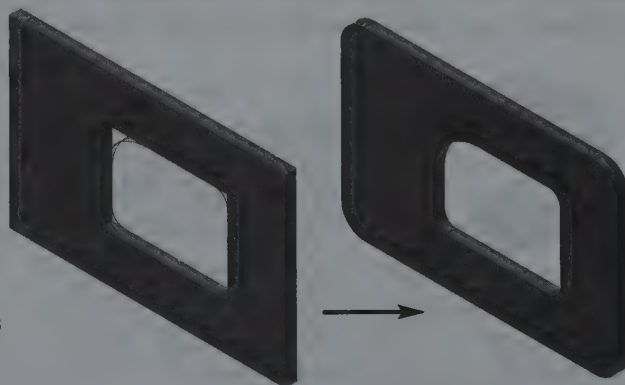
Corner Chamfers

A *corner chamfer* is commonly used to remove sharp points and edges or to apply required angular geometry. Access the **Corner Chamfer** tool to place corner chamfers using the **Corner Chamfer** dialog box. You can add all of the chamfers shown in **Figure 16-14**, for example, using a single corner chamfer operation. All of the chamfers are then displayed as a single feature in the browser.

Ribbon

Sheet Metal > Modify

Corner Chamfer

> **NOTE**
>
> The **Corner Chamfer** tool creates one or more corner chamfers of the same size. You must reuse the tool to create a corner chamfer of a different size. All corner chamfers placed during one operation make up a single feature in the browser.

To create a corner chamfer, select the appropriate chamfer method button. **Figure 16-15** shows how each method defines the size and shape of the corner chamfer. Use an approach similar to that for creating chamfers using the **Chamfer** tool. When you apply the **Distance and Angle** option, the **Edge** button allows you to select the edges from which the angle measures, actually a thin sheet metal face. The **Corners** button allows you to select the chamfered corners from which the chamfer distance measures. See **Figure 16-15**. Pick the **Apply** button to create the feature and remain in the **Corner Chamfer** tool, or pick the **OK** button to create the feature and exit.

> **NOTE**
>
> Deselect selected edges by holding down [Ctrl] and picking the edges to remove from the operation.

Exercise 16-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-8.

Figure 16-14.
Use the **Corner Chamfer** dialog box to apply corner chamfers. You must reuse the tool to create chamfers of a different size.

Two Distance button
Distance and Angle button
One Distance button
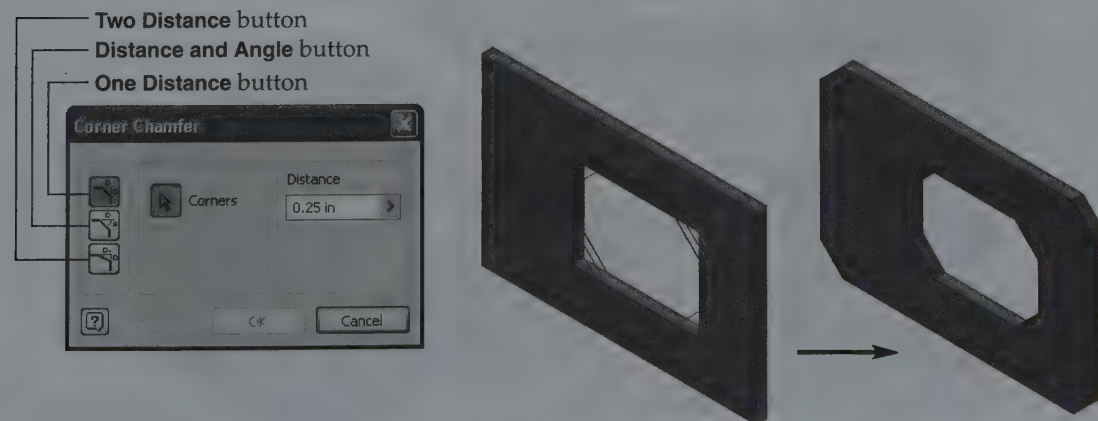
Figure 16-15.
The three methods of defining corner chamfers using the **Corner Chamfer** dialog box.

One Distance   Distance and Angle   Two Distances

Selected edge corresponds to the distance

Selected corner (removed by chamfer) corresponds to the angle
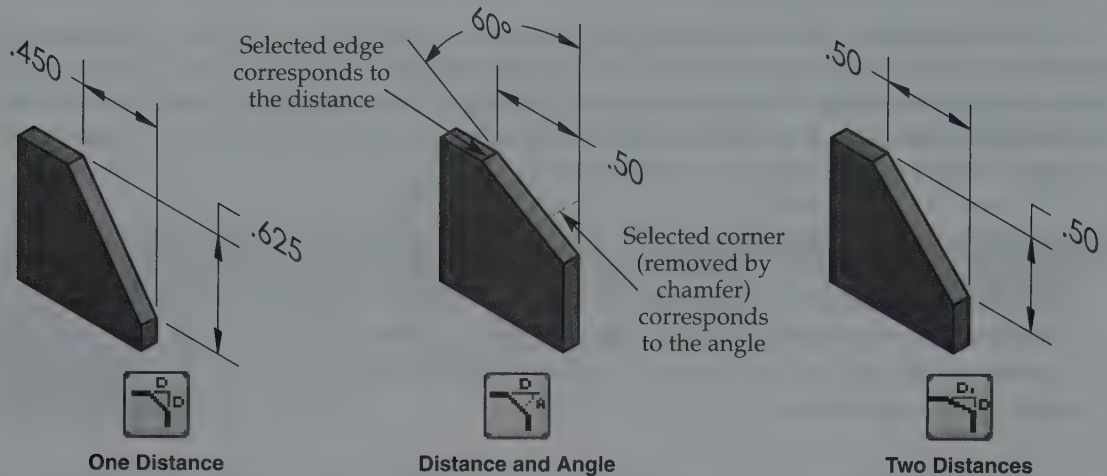
Corner Seam Tool

corner seams: Features that add or remove material to form a gap at sheet metal part corners.

corner rip: A feature that opens closed, usually square, corners.

Ribbon
Sheet Metal > Modify
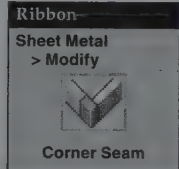
Corner Seam

Seams at intersecting, perpendicular, or coplanar sheet metal corners are usually created during part development using sheet metal feature tools. However, it may be necessary to modify a *corner seam*, or add a corner seam in the form of a *corner rip*, such as when building a sheet metal part from a converted non-sheet metal part. Access the **Corner Seam** tool to modify or add corner seams using the **Corner Seam** dialog box. See **Figure 16-16**.

Nonplanar Feature Corner Seams

To modify a corner seam between nonplanar features, select the **Seam** radio button, and then use the **Edges** button to pick edges. A preview of the operation appears, showing material to add and/or remove to form a gap, depending on the selected edges and existing features. The **Maximum Gap Distance** radio button is active by default and is intended to replicate a gap measured using a groove or gap gage. The **Face/Edge Distance** option applies the method of corner seam placement used in versions of

Figure 16-16.
Adding a corner seam between nonplanar features using the **Corner Seam** dialog box.

Inventor and Its Applications 2010
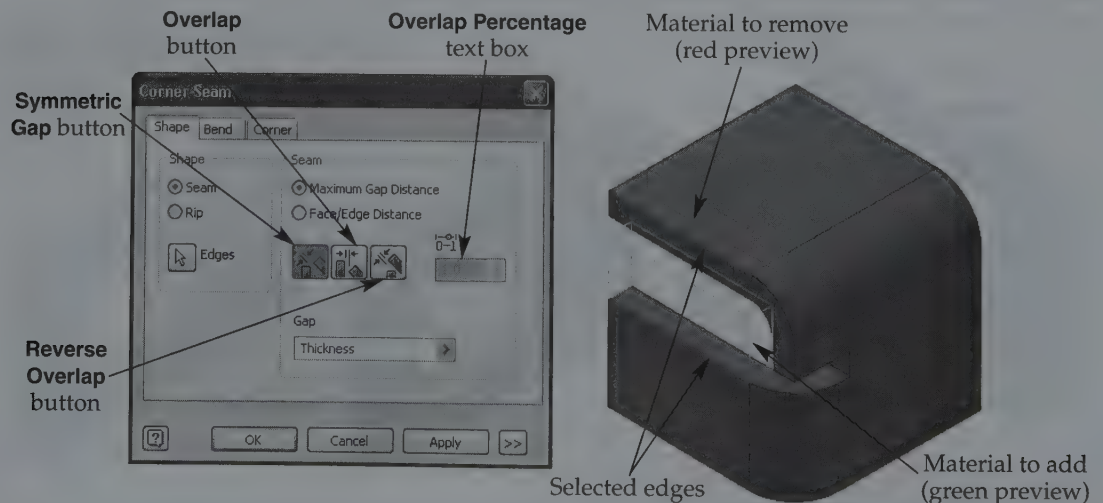
Figure 16-17.
Corner seam options available for placing a seam between nonplanar features.

**Maximum Gap Distance
Symmetric Gap**
(Trim To Bend
relief shown)

**Maximum Gap Distance
Overlap**

OR

**Face/Edge Distance
Overlap**

**Face/Edge Distance
No Overlap**

**Maximum Gap Distance
Reverse Overlap**

OR

**Face/Edge Distance
Reverse Overlap**
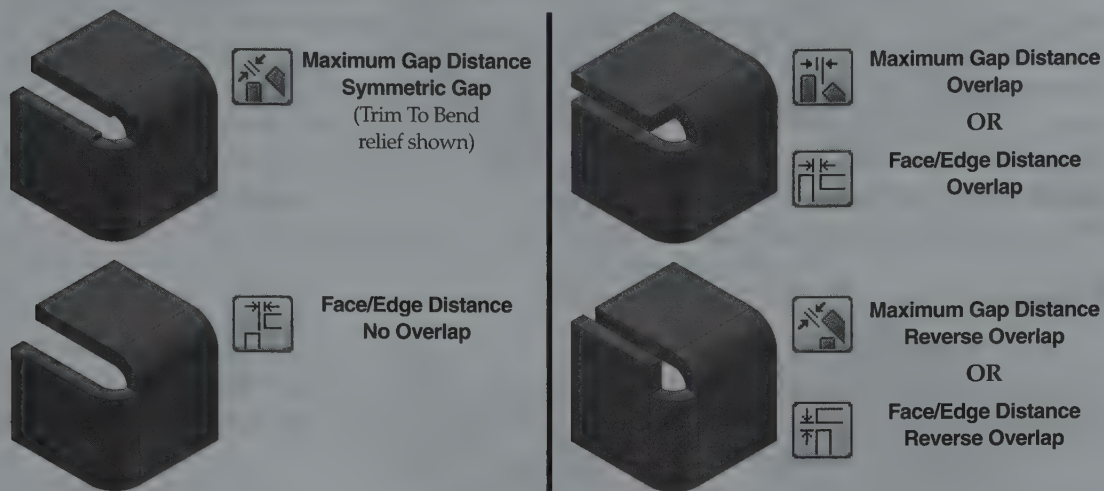
Inventor prior to 2008. In this method, the gap is measured from the face next to the first selected edge to the second selected edge. **Figure 16-17** shows an example of each seam option applied to the part shown in **Figure 16-16**.

The **Percent Overlap** text box becomes enabled when you apply **Overlap** or **Reverse Overlap** corner seams. If necessary, enter a value from 0 to 1 to adjust the overlap. See **Figure 16-18**. Adjust the seam gap using the **Gap** text box only if you want to override the default seam opening with a value greater than 0.

Figure 16-18.
Examples of overlap percentages. You can also adjust reverse overlap percentages.

Isometric view   Side view
Percent Overlap = 1

Isometric view   Side view
Percent Overlap = .5

Isometric view   Side view
Percent Overlap = 0
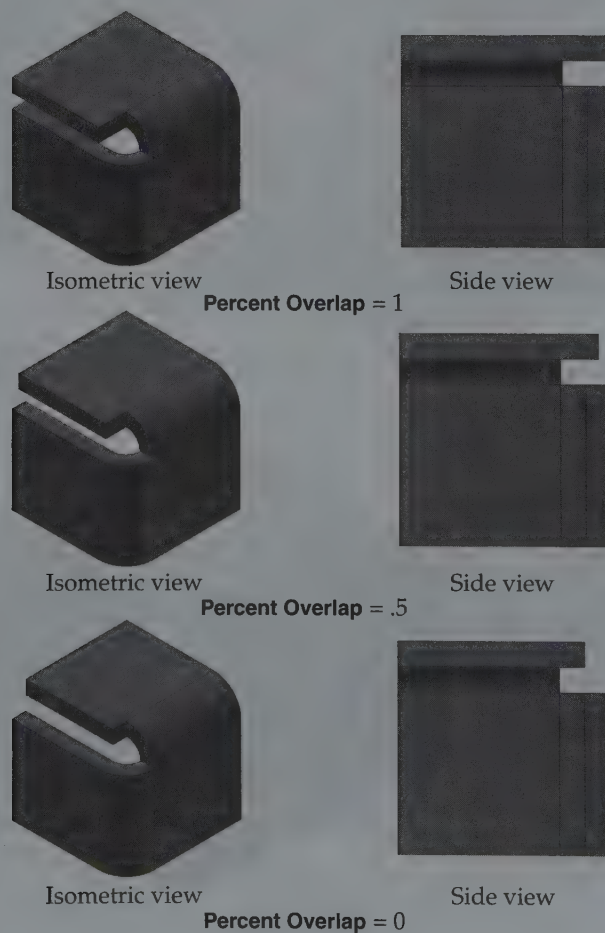
Coplanar Feature Corner Seams

To modify a corner seam between coplanar features, select the **Seam** radio button, and then use the **Edges** button to pick edges. **Figure 16-19** shows two possible pairs of selected edges. A preview of the operation appears, showing material to add and/or remove to form a gap, depending on the selected edges and existing features. Use the default **Maximum Gap Distance** option or pick the **Face/Edge Distance** radio button. **Figure 16-20** shows examples of each seam option applied to each pair of selected edges shown in **Figure 16-19**. Adjust the seam gap using the **Gap** text box only if you want to override the default seam opening with a value greater than 0.

Corner Rips

Pick the **Rip** radio button to rip an intersecting, often square, corner. See **Figure 16-21**. Rips may be required during sheet metal part development, but they are most commonly used when creating a sheet metal part from a standard part. The

Figure 16-19.
Adding a corner seam between coplanar features.

45 Degrees button

Overlap Percentage text box

Selected edges

Material to add

Overlap button

Reverse Overlap button

A

Selected edges

B
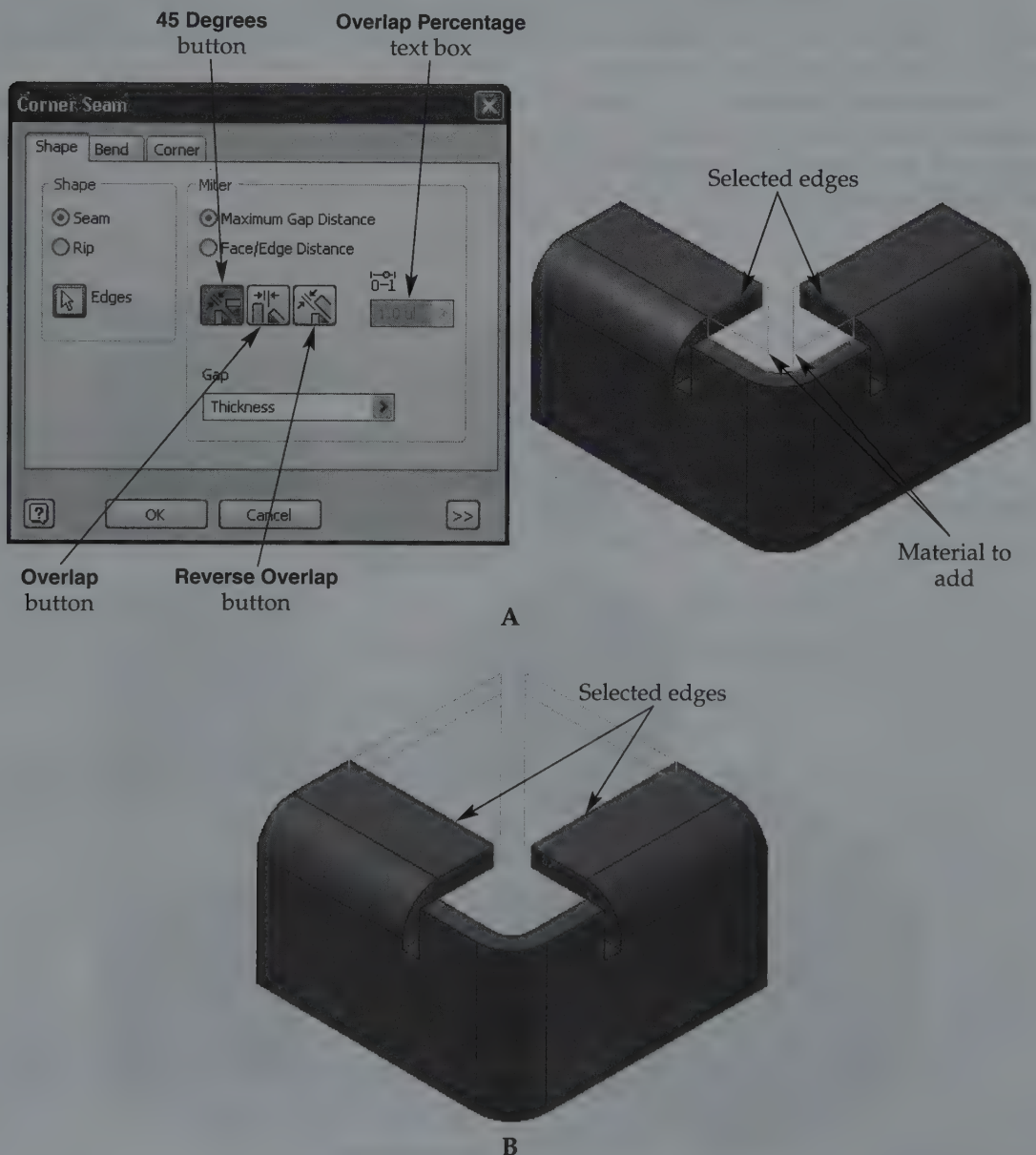
Figure 16-20.
Corner seam options
available for placing
a seam between
coplanar features.

Maximum Gap Distance
45 Degrees
OR
Face/Edge Distance
Symmetric Gap

Maximum Gap Distance
Overlap
OR
Face/Edge Distance
Overlap

Maximum Gap Distance
Reverse Overlap
OR
Face/Edge Distance
Reverse Overlap

Figure 16-21.
Creating a corner rip. This part can now unfold into a flat pattern.

Pick to complete a corner rip

Material to remove

Corner Seam

Shape | Bend | Corner

Shape
○ Seam
◉ Rip
⊡ Edges

Seam
○ Maximum Gap Distance
◉ Face/Edge Distance

Gap
Thickness

Cancel   Apply   <<

Measure Gap

Extend Corner
◉ Aligned
○ Perpendicular

Selected edge is
interior corner not
visible in this view
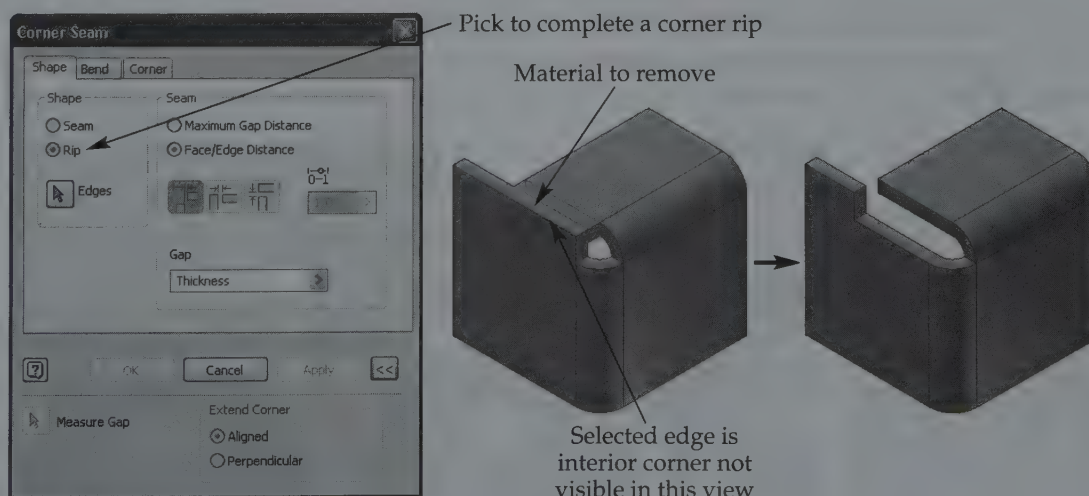
Edges button is active, allowing you to pick existing feature edges. The **Face/Edge Distance** radio button is automatically activated. Select the appropriate overlap option or pick the **Maximum Gap Distance** radio button, which only allows you to create a symmetric gap. Adjust the rip gap using the **Gap** text box only if you want to override the default seam opening with a value greater than 0.

Additional Options and Finalizing

Pick the **More** button to access additional corner seam options. The **Measure Gap** button is available when you create a 45° corner miter. Pick the **Aligned** radio button to align the selected face edges by projecting the first face edge to the second face edge. Choose the **Perpendicular** radio button to project the first selected face edge perpendicular to the second selected face edge.

The settings in the **Bend** and **Corner** tabs are the same as those located in the **Style and Standard Editor**. Adjust the settings only if you want to override the default sheet metal parameters. Pick the **Apply** button to create the feature and remain in the tool, or pick the **OK** button to create the seam or rip and exit.

> **NOTE**
>
> To change a corner seam into a bend, right-click on the corner seam in the browser and select **Change to Bend**. The corner seam is removed and the **Bend** dialog box appears.

Exercise 16-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-9.

Rip Tool
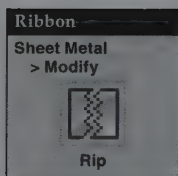
The **Rip** tool allows you to use the **Rip** dialog box, shown in **Figure 16-22**, to add a seam opening when one does not form during sheet metal part development, or to remove a face. Unless you plan to remove a face, you must first sketch a point or two points to indicate the extents of the rip. You can select a curve endpoint or center point

Figure 16-22.
Using the **Rip** dialog box to add a seam on a face using a single sketch point.

Selected sketch point (single
Point, Center Point sketched on rip face)

Selected rip face
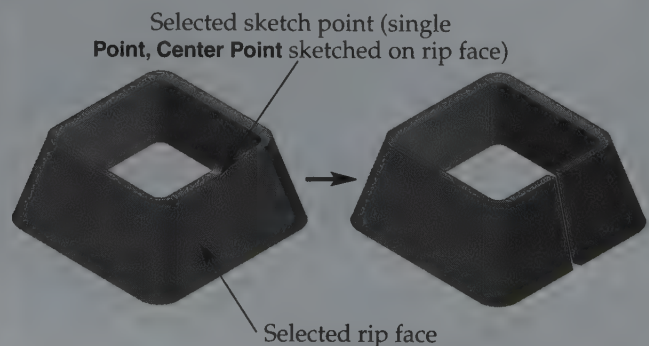
or a point sketched using the **Point, Center Point** tool. Depending on feature geometry, you may be able to create a rip perpendicular to an edge at a single sketch point, using the **Single Point** rip type, as shown in **Figure 16-22**. Use the **Rip Face** button to select the face to receive the seam. Then pick the **Sketch Point** button and select the sketched point on the face.

To create a seam that is not perpendicular to an edge, or a nonlinear seam, sketch two points and use the **Point to Point** rip type. See **Figure 16-23**. Use the **Rip Face** button to select the face to receive the seam. Select the **Start Point** button and pick the first sketched point on the face, and then select the **End Point** button and pick the second sketched point. The **Single Point** and **Point to Point** options include **Flip** and **Both Sides** buttons that allow you to adjust where the rip occurs. If necessary, you can override the default seam size using the **Gap Value** text box.

To remove a face, select the **Face Extents** rip type, followed by the face to remove. See **Figure 16-24**. Removing a face is common when part development forces an unnecessary face, or to create a seam at a bend. Pick the **Apply** button to apply the rip and remain in the tool, or pick the **OK** button to create the rip and exit.

Figure 16-23.
Ripping a seam from one sketch point to another.

Selected rip face

Selected endpoint
(projected curve endpoint
sketched on rip face)

Selected start point
(projected curve endpoint
sketched on rip face)
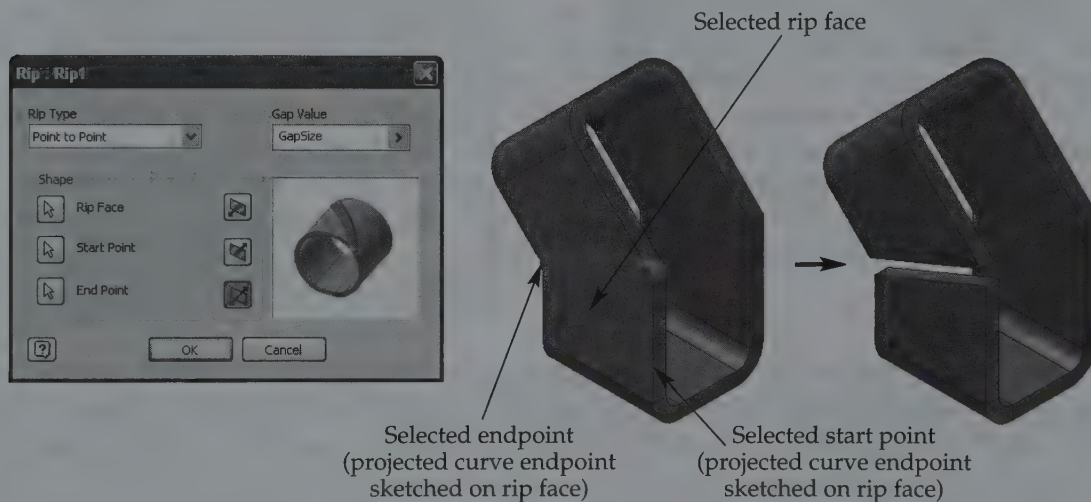
Figure 16-24.
Removing a face using **Face Extents** option of the **Rip** tool. In this example, a bend is removed to create a seam.

Selected rip face

While developing a sheet metal part, you may find it necessary to create features by referencing the flat pattern, because adding these features to the folded part may be too difficult or create incorrect folded or flat pattern geometry. You can unfold a sheet metal part to create a flat pattern, as described later in this chapter. However, Inventor does not retain features added to a flat pattern when you return to the folded model. The solution is to use the **Unfold** and **Refold** tools.

> **NOTE**
>
> You may find the **Unfold** and **Refold** tools more effective than using the **Project Flat Pattern** tool for some applications, such as when working with complex model geometry, or when a sketch requires a work plane.

Unfolding
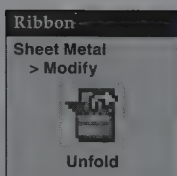
Access the **Unfold** tool to unfold a folded modeling using the **Unfold** dialog box. See **Figure 16-25**. Use the **Stationary Reference** button to select a face that remains fixed during the unfold operation. All bends that can unfold are highlighted in a different color than the stationary reference. Use the **Bends** button to pick individual bends to unfold, or select the **Add All Bends** button to unfold the entire model, if possible.

Next, use the **Sketches** button to select unconsumed sketches that were not sketched on the stationary reference to project onto the unfolded model as a copy. The selected **Parent is Visible** check box maintains visibility of the parent sketch, until you turn off the visibility. Pick the **Apply** button to unfold the part and remain in the tool, or pick the **OK** button to unfold the part and exit.

Refolding

Add features to an unfolded part as necessary. When you are finished, return to the previous folded state by right-clicking on the Unfold feature in the browser and selecting **Refold Feature**. See **Figure 16-26A**. Any features added to the unfolded model

Figure 16-25.
Use the **Unfold** tool to unfold a part temporarily. Once the part is unfolded, you can add features positioned better according to the flat pattern.
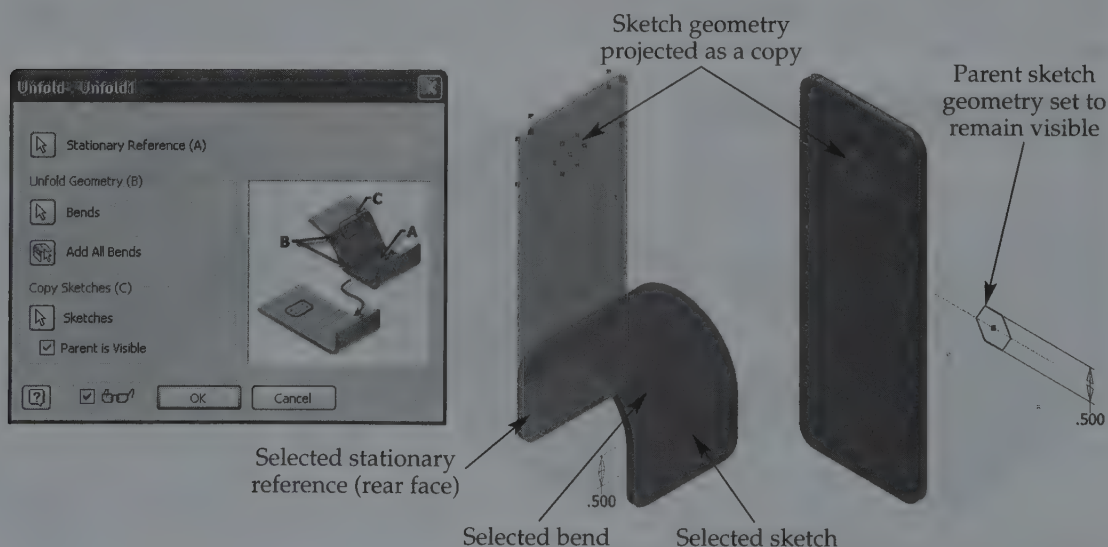
Figure 16-26.
A—Adding features in the unfolded state and then refolding to return to the original model.
B—Use the **Refold** tool to refold a model with a copy of an unconsumed sketch or to refold in
a different format.

Cut using
additional sketch

Cut using sketch
projected from
folded model

Sketch plane
for additional
sketch

4.000

1.000

A

Selected stationary
reference

Refold: Refold3

Stationary Reference (A)

Refold Geometry (B)

Bends

Add All Bends

Copy Sketches (C)

Sketches

Parent is Visible

OK   Cancel

Selected bend

B
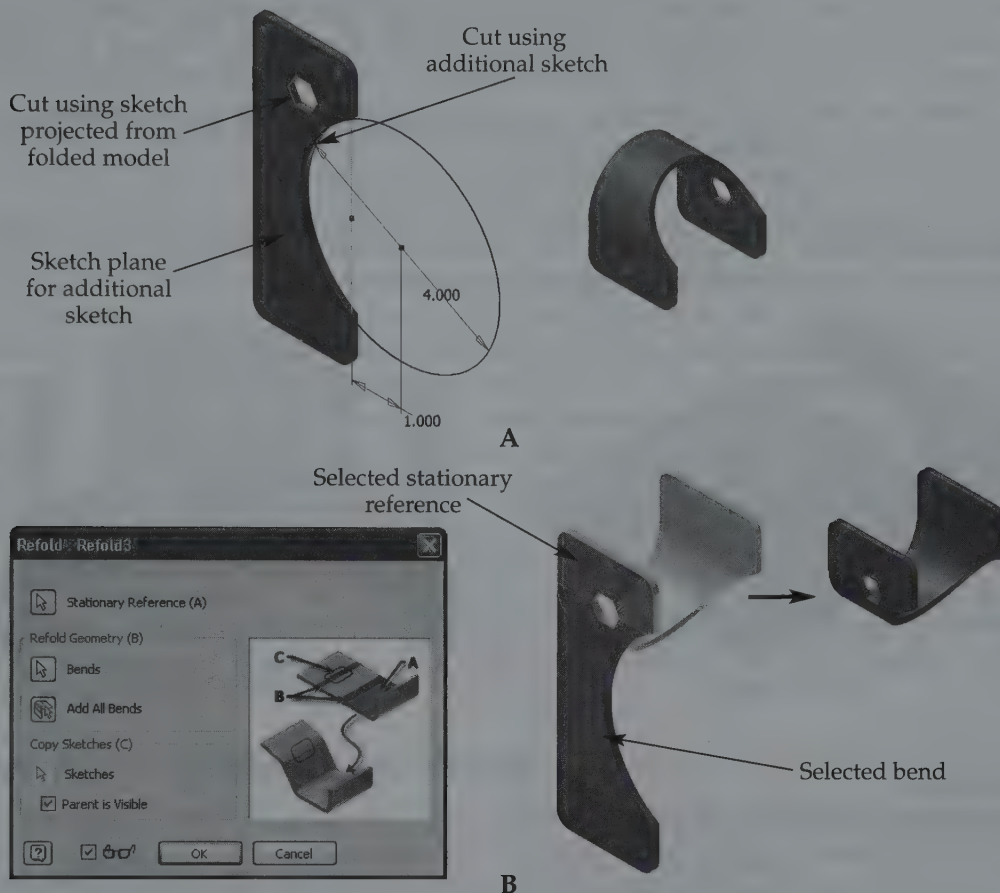
are folded with the model. Unconsumed sketches added to the unfolded part remain
with the unfolded feature.

To include unconsumed sketches during the refold or to fold the model differently,
access the **Refold** tool to display the **Refold** dialog box. The **Refold** tool includes the
same options as the **Unfold** tool, but is a reverse of the **Unfold** tool. **Figure 16-26B** shows
an example of using the **Refold** tool to refold a model into a different orientation.

Exercise 16-10

Access the Student Web site (www.g-wlearning.com/CAD) and
complete Exercise 16-10.

Flat Patterns

Once you create a folded sheet metal part, or at any time during sheet metal part development, you can use the **Flat Pattern** tool to display the flat pattern. See **Figure 16-27**. The flat pattern appears in a separate work environment, as shown in **Figure 16-28**. The modeling and sheet metal tools provided in the flat pattern work environment allow you to view and adjust the flat pattern as needed. Changes made to the flat pattern are not reproduced in the folded model.

Figure 16-27.
A folded sheet metal part and the corresponding flat pattern.

Folded Model
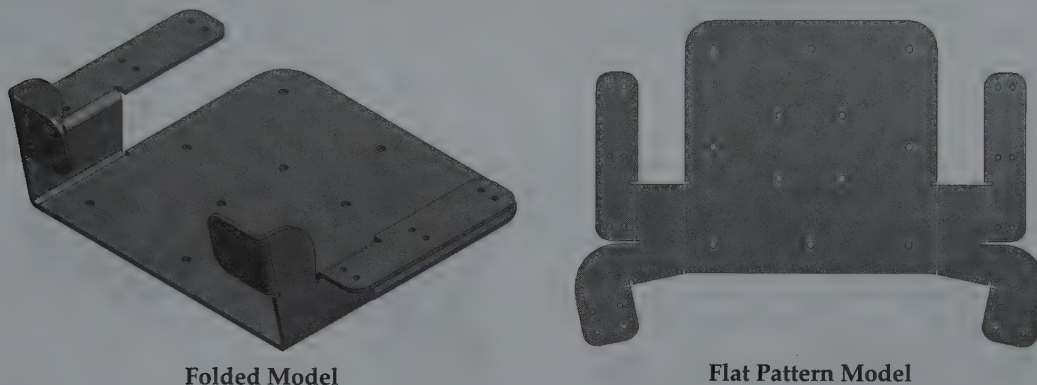
Flat Pattern Model

Figure 16-28.
The flat pattern work environment.

Double-click to return to the folded model

Pick to return to the folded model
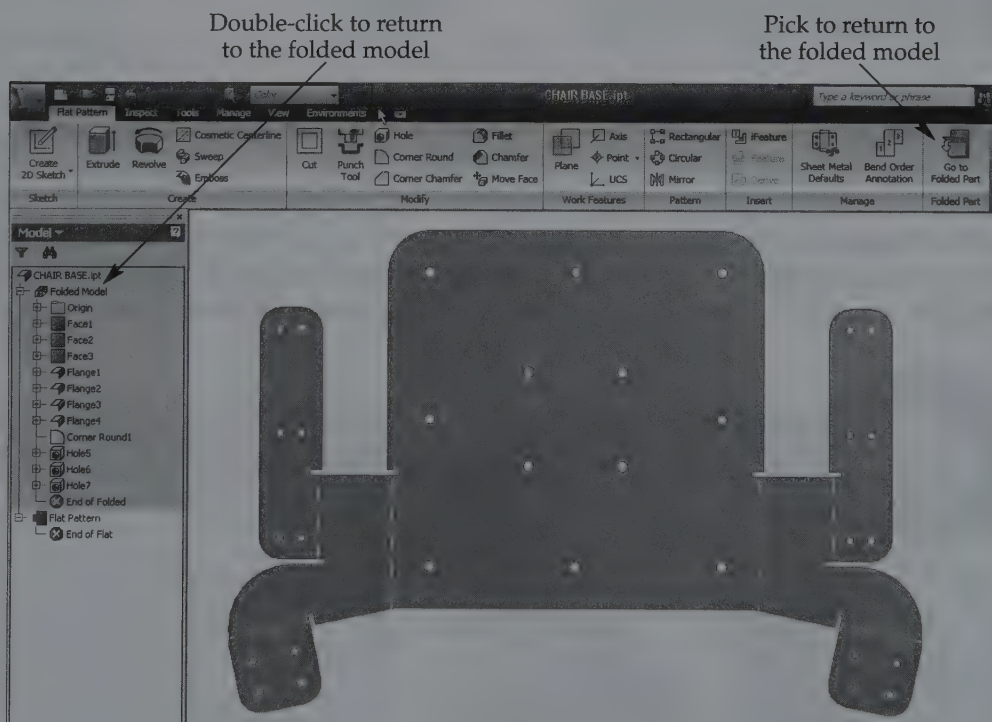
You can toggle between the folded and flat model as needed. To re-enter the folded model work environment, use the **Folded Part** tool, or double-click or right-click on Folded Model in the browser and select **Edit Folded**. To re-enter the flat pattern work environment, use the **Flat Pattern** tool, or double-click or right-click on Flat Pattern in the browser and select **Edit Flat Pattern**. The flat pattern remains in the file until you delete it by right-clicking on Flat Pattern in the browser and selecting **Delete**.

Ribbon
Sheet Metal
> Flat Pattern

Go to Folded Part

Editing the Flat Pattern Definition

Right-click on Flat Pattern in the browser and select **Edit Flat Pattern Definition** to display the **Flat Pattern** dialog box. See **Figure 16-29**. To observe changes to settings in the **Flat Pattern** dialog box, you must select the **Apply** button to remain in the dialog box, or select the **OK** button to display the changes and exit the tool.

The **Orientation** tab allows you to adjust the flat pattern orientation by aligning, or rotating, the flat pattern in reference to an axis. Use the **Alignment Axis** select button to pick a curve to specify where alignment occurs. Pick the **Flip Alignment Axis** button to reverse the alignment axis. Select the **Align Horizontal** radio button to align the flat pattern with the horizontal axis, or choose the **Align Vertical** radio button to align the flat pattern with the vertical axis. Select the **Flip to opposite face** button to specify the opposite flat pattern face as the base face.

Right-click on the **Default** orientation and select the appropriate option to create, rename, activate, delete, or save changes to the active orientation. You can also use the **Save** button to save changes to the active orientation. Adjust the settings in the **Punch Representation** and **Bend Angle** tabs only if you want to override the default sheet metal parameters.

Cosmetic Centerlines and Extents

Bend centerlines and extents appear by default in the flat pattern. Use the representations to view bend parameters and take measurements as needed. Appropriate bends usually form during folded model development. If you cannot build a certain folded feature, but still require bend content in a flat pattern drawing, sketch a bend centerline across an entire flat pattern face. Then access the **Cosmetic Centerline** tool to convert the line using the **Cosmetic Centerlines** dialog box. Select the sketched line and specify bend parameters. The default **Press Break** option allows you to indicate a bend, similar to using the folded part **Fold** tool. The **None** option creates a centerline, and the **Crease** option allows you to identify a crease or score line.

Ribbon
Flat Pattern
> Create

Cosmetic
Centerline
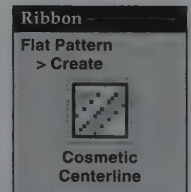
Figure 16-29.
Use the **Flat Pattern** dialog box to adjust flat pattern definition options.
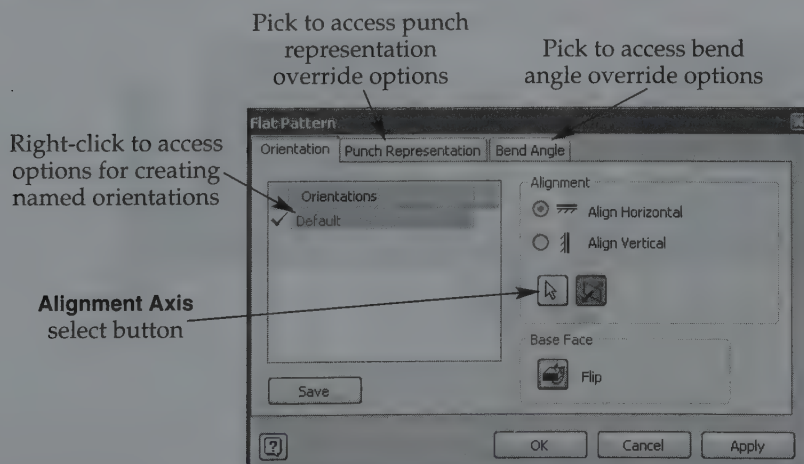
Bend Order Annotation
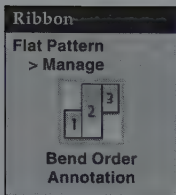
The sequence, or order, in which bends occur, is usually essential to sheet metal part forming. Sometimes a flat pattern drawing includes the order, especially when a bend table documents bend parameters. See **Figure 16-30**. Inventor automatically assigns bend order based on folded model and flat pattern geometry. Use the **Bend Order Annotation** tool in the flat pattern work environment of the sheet metal part file to view and adjust bend order.

Glyphs appear, indicating the default order. See **Figure 16-31A**. For most applications, you can make changes to the bend sequence by right-clicking and selecting a reorder option. Select **Directed Order** to pick the first and last bend glyphs, allowing Inventor to attempt to calculate the correct order. Choose **Sequential Order** to pick glyphs in sequential order until the correct override order appears. See **Figure 16-31B**. You also have the option of picking a glyph to edit the glyph, including duplicating numbers, using the **Bend Order Edit** dialog box. To eliminate overrides, right-click and select **Remove All Overrides**. When you are finished, right-click and select **Finish Bend Order**.

Figure 16-30.
An example of a drawing with bend parameters extracted from the model to form a bend table.

BEND TABLE			
BEND ID	DIRECTION	ANGLE	RADIUS
1	DOWN	90	.25
2	DOWN	90	.25
5	DOWN	90	.25
3	UP	90	.25
4	UP	90	.25
6	DOWN	90	.25
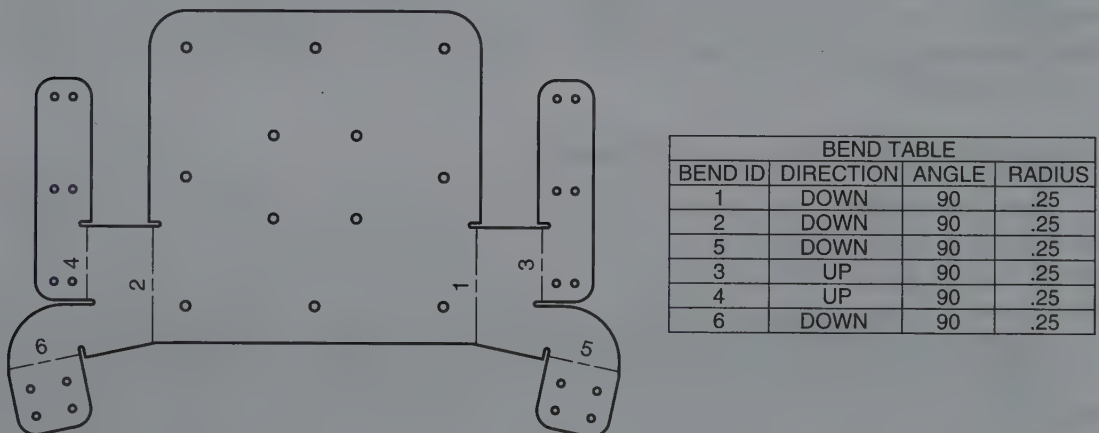
Figure 16-31.
A—Glyphs representing the default bend order. B—Reordering glyphs sequentially until the correct sequence appears.

Select glyphs in sequential order to override

A

B
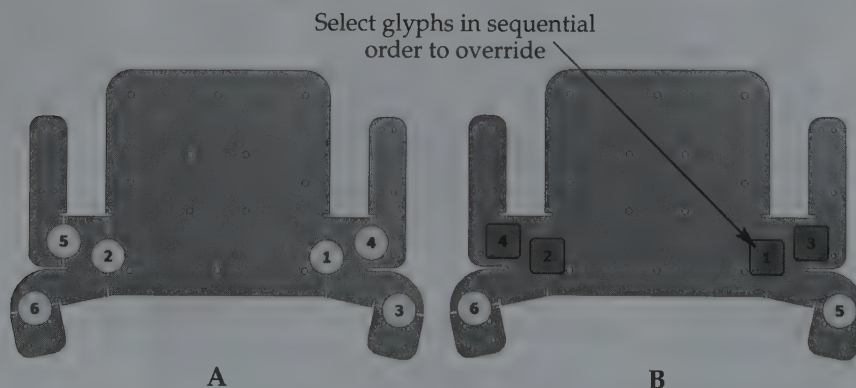
Viewing Extents

Right-click on Flat Pattern in the browser and select **Extents...** to show the **Flat Pattern Extents** display box. See **Figure 16-32**. Use this function to view the overall material width, length, and area needed to create the part. You cannot edit the length and width in the **Flat Pattern Extents** display box, but you can use measuring tools to view pattern measurements.

> **NOTE**
>
> Complex features, such as embossed iFeatures, may remain as shown in the folded model, while other features unfold.

> **PROFESSIONAL TIP**
>
> Right-click on Flat Pattern and select **Save Copy As...** to open the **Save Copy As** dialog box, which allows you to save a copy of the flat pattern as an SAT, DWG, or DXF file.

Exercise 16-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 16-11.

Figure 16-32.
Right-click on Flat Pattern in the browser and pick **Extents...** to display the width, length, and area of the flat pattern.
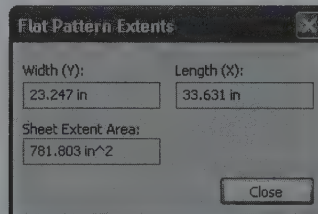
Flat Pattern Extents

Width (Y):
23.247 in

Length (X):
33.631 in

Sheet Extent Area:
781.803 in^2

Close

Chapter Test

1. What is a bend line?
2. How can you quickly change a bend into a corner seam?
3. Explain how to create a cut that wraps around bends and faces that are not coplanar to the sketch plane.
4. Explain the basic function of a sheet metal punch.
5. What is the difference between inserting an iFeature and using the **PunchTool**?
6. When might you use a punch ID?
7. Describe the "dangling geometry" condition and explain its cause.
8. Describe a corner round.
9. What are corner chamfers?
10. Briefly describe corner seams.
11. What is a corner rip?
12. Which tool allows you to create a seam opening in a model?
13. When you unfold a sheet metal part to create a flat pattern, Inventor does not retain features added to a flat pattern when you return to the folded model. Name the tools that provide the solution to this situation.
14. Name the tool used to display a flat pattern after it has been created.
15. What does Inventor use as a basis to assign bend order?

Problems

Instructions: Follow the specific instructions for the following problem.

1. Open file P15-4.ipt and save it as P16-1.ipt. In the P16-1.ipt file, open a sketch on the face shown and create the sketch geometry. Sketch rectangles collinear to the contour flange edges to fully constrain the sketch. Cut the profiles shown through all to create the final part. Resave the part.

2–6 Instructions:

- *Create sketches of the following objects.*
- *Develop sketch geometry from the projected center point.*
- *Infer as many geometric constraints as possible and appropriate.*
- *Add geometric constraints as appropriate, and use equal constraints for like objects not dimensionally constrained in the problem figure.*
- *Use the information in the status bar to create objects the approximate size given by the dimensional constraints.*
- *Add the dimensional constraints shown.*
- *Add as much information as possible to the **iProperties** dialog box. Assign the specified material and color to the part.*
- *Follow the specific instructions for each problem to create the features.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

2. **Title:** HOUSING
Units: Inch
Template: Sheet Metal-IN.ipt
Part Number: IAA-039-01
Project: HOUSING
Material: According to sheet metal rule
Color: Galvanized (texture)
Sheet Metal Rule: 11 GAGE STEEL
Save as: P16-2.ipt
Specific Instructions:

A. Create a 4″ × 6″ face and add 2″ flanges around the face as shown. The flanges should offset toward the face and occur in a positive direction. Modify each of the corners with a **Maximum Gap Distance – Symmetric Gap** seam and a round bend relief as shown. Add two flanges to the 4″ long sides first.

B. Add two 1″ flanges as shown. The flanges should offset toward the base flanges.

C. Place four .25″ holes on the flanges as shown. Use horizontal and vertical constraints to pattern the hole centers.

D. Sketch and constrain a circle on the face shown, and cut the circle across the bend of the specified flange.

E. Sketch and constrain a rectangle on the face shown, and cut the rectangle across the bend of the specified flange.

The final part should look like the part shown.

Tangent constraint

2.7600   1.5000

.3750

.3750

.5000

.5000

3.0000

1.5000

1.2600
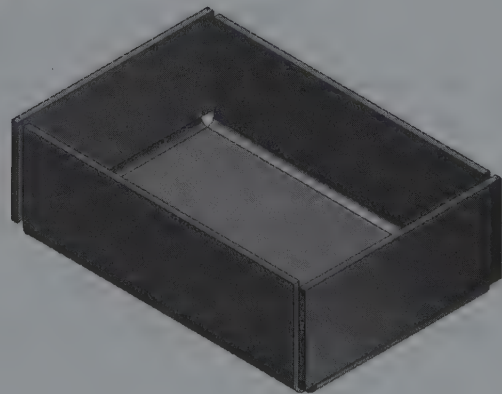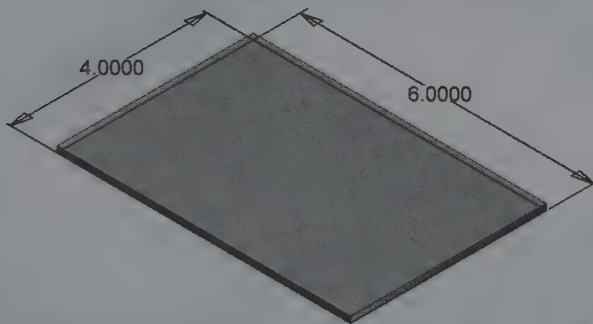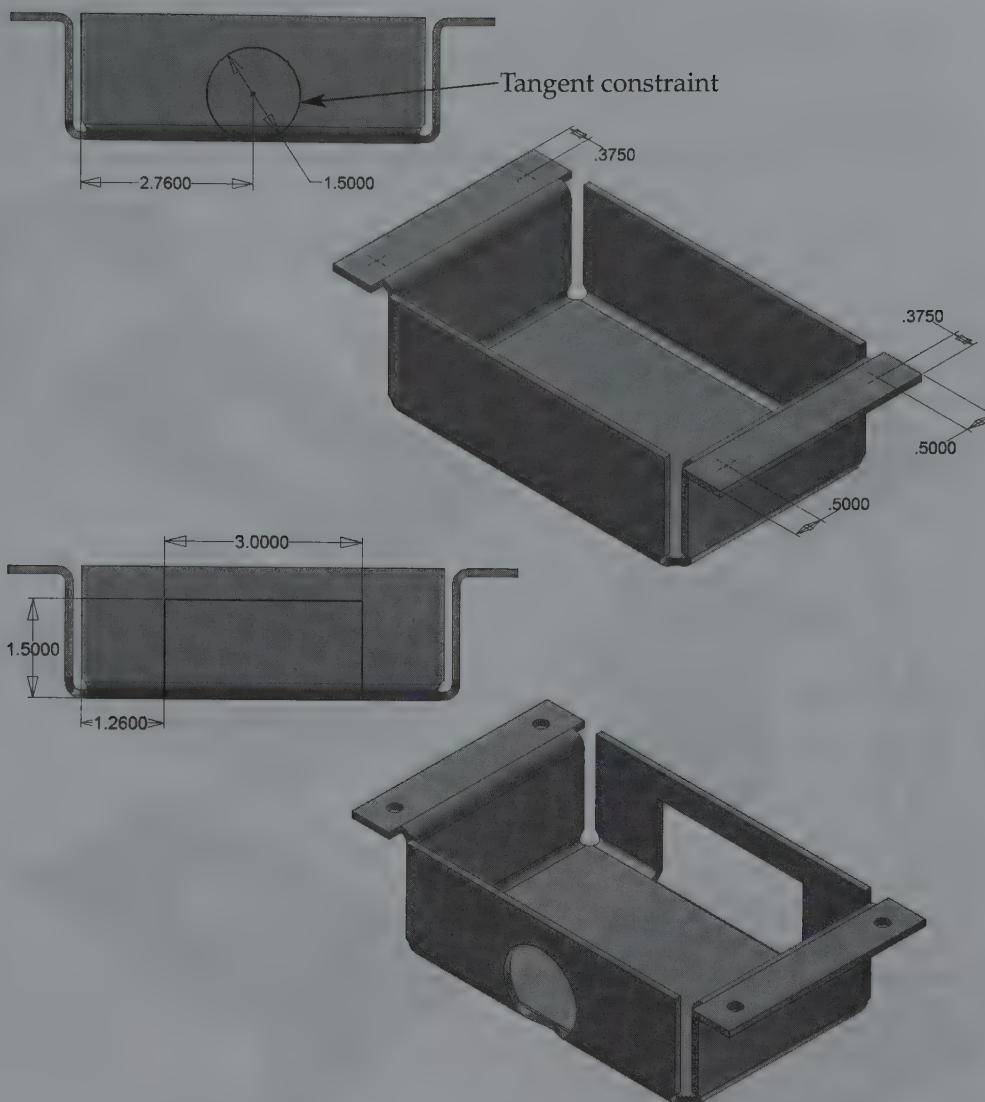
3. **Title:** LOUVER
   Units: Metric
   Template: Sheet Metal-mm.ipt
   Part Number: IAA-040-00
   Project: LOUVERED COVER
   Material: According to sheet metal rule
   Color: As Material
   Sheet Metal Rule: 0.5 430
   Save as: P16-3.ipt
   Specific Instructions:
   A. Create a 50 mm × 100 mm face as shown.
   B. Sketch the rectangle and **Point, Center Point** on the face as shown for reference. Cut the rectangle through the face and place R12 mm corner rounds as shown.
   C. Open a sketch on the inside of the cut and sketch the geometry shown. You may want to use **Slice Graphics** for clarity. This sketch is a sweep profile.
   D. Open a new sketch on the top of the face and project the geometry shown. This sketch is a sweep path.
   E. Access the **Part Features** tools and create a sweep feature using Sketch3 as the profile and Sketch4 as the path. The sweep should look like the feature shown.

F. Using the **Create iFeature** dialog box, create an iFeature from the cut, corner round, and sweep. Name the iFeature Louver, and specify thickness, length, and width size parameters. Save the iFeature in a location of your choice for future access.

The final part should look like the part shown.

Sketch profile

27.5
19
6
12
45
22.5
100
50

12.5
R11.5
0.5
12
R11
0.5

12.5
11.6
0.5
12
11
0.5
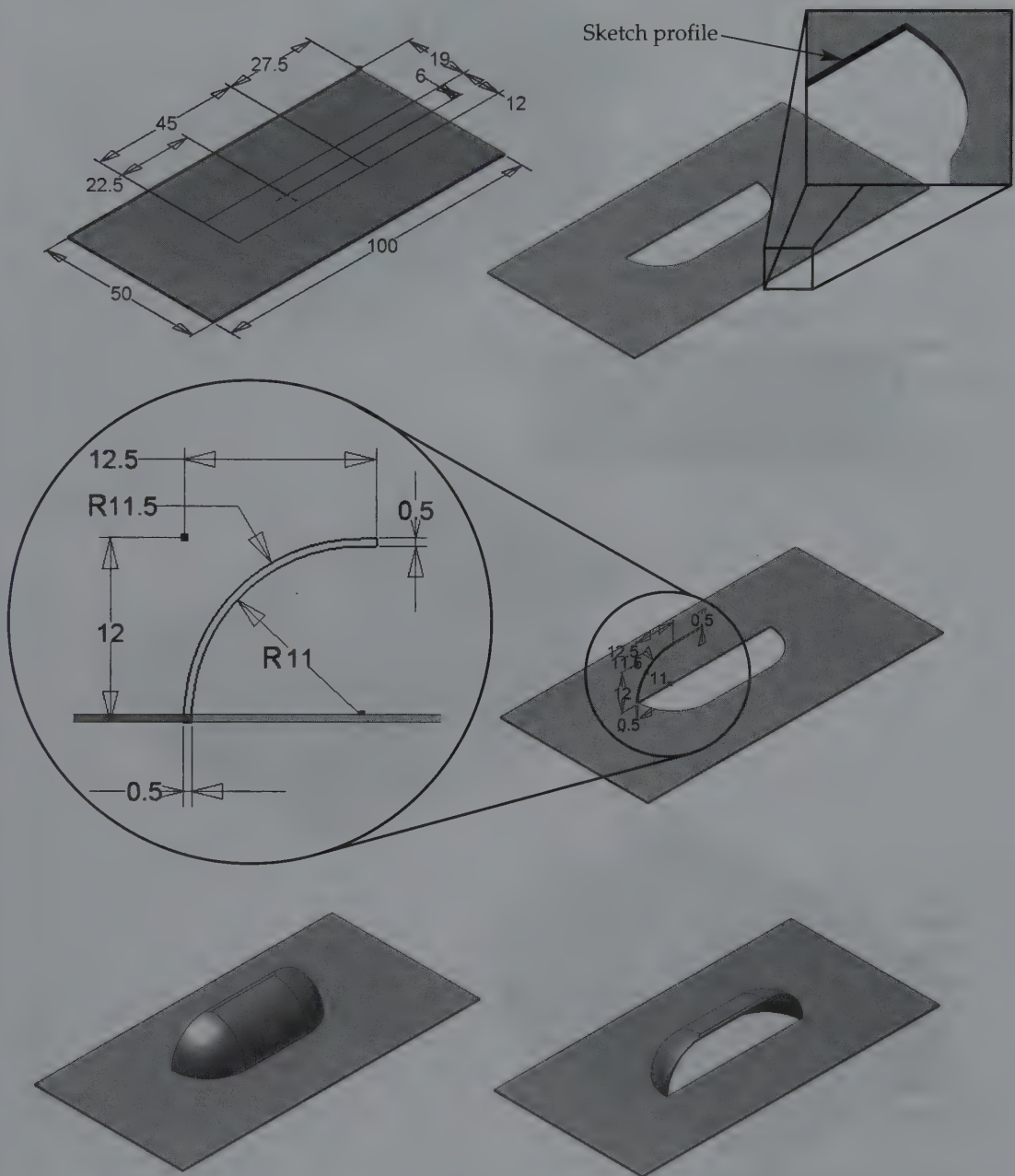
4. **Title:** LOUVERED COVER
   Units: Metric
   Template: Sheet Metal-mm.ipt
   Part Number: IAA-040-01
   Project: LOUVERED COVER
   Material: According to sheet metal rule
   Color: As Material
   Sheet Metal Rule: 0.5 430
   Save as: P16-4.ipt
   Specific Instructions: Create a 300 mm × 400 mm face and add 25 mm flanges to the 300 mm edge as shown. Add single hems to the 400 mm edge as shown. Sketch a **Point, Center Point** on the top of the face as shown. Insert the louver iFeature created in P16-2.ipt on the **Point, Center Point**. Use the **Rectangular Pattern** tool to space louvers equally as shown. The final part should look like the part shown.

Hems

Flanges

25   50
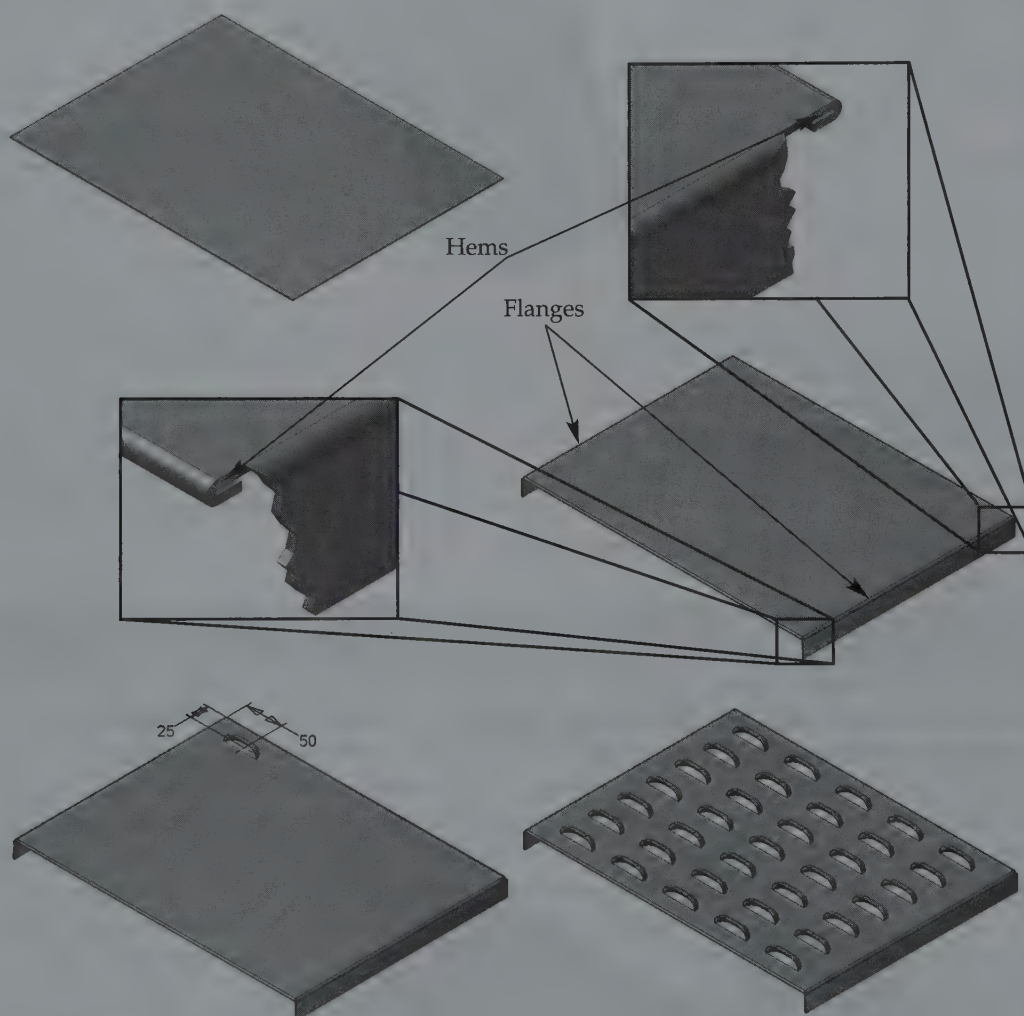
5. **Title:** BOTTLE HOLDER
   Units: Inch or Metric
   Template: Sheet Metal-IN.ipt or Sheet Metal-mm.ipt
   Part Numbers: IAA-038-01
   Project: BOTTLE HOLDER
   Material: According to sheet metal rule
   Color: White
   Sheet Metal Rule: Create an appropriate rule
   Save as: P16-5.ipt
   Specific Instructions: Use a single contour flange and fold to create a refrigerator two-liter bottle holder similar to the part shown.
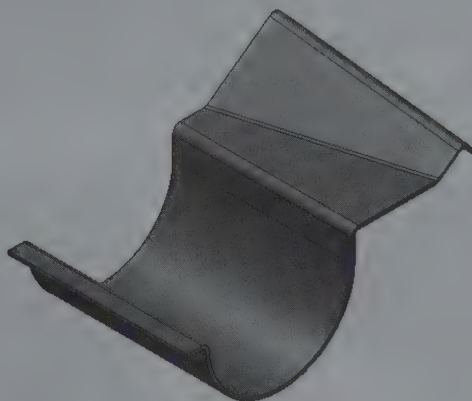
6. **Title:** SEAT BASE
   Units: Inch
   Template: Sheet Metal-IN.ipt
   Part Number: IAA-041-01
   Project: SEAT
   Material: According to sheet metal rule
   Color: Default
   Sheet Metal Rule: 4 GAGE 6061
   Save as: P16-6.ipt
   Specific Instructions: Create a sheet metal seat base similar to the seat base shown. When finished, use the **Flat Pattern** tool to view the seat base flat pattern. Save the flat pattern as P16-6.sat.
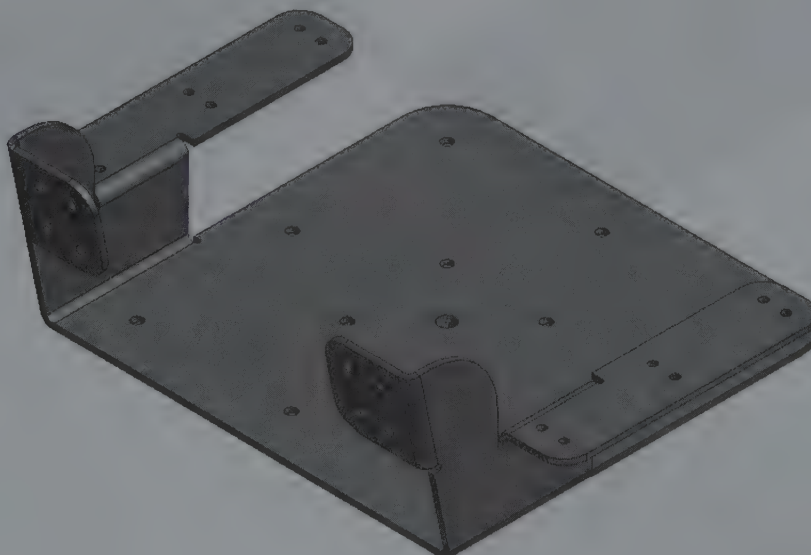
Introduction to Assemblies

17

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Describe methods of producing assembly models.
✓ Insert components to build assemblies.
✓ Assemble components using constraints.
✓ Drive constraints to view and animate assembly movement.

Use an *assembly* file to bring multiple parts and *subassemblies*, known as *components*, together to form an assembly model. See **Figure 17-1**. Assembly constraints allow you to assemble components. This chapter focuses on beginning an assembly, inserting components, and applying assembly constraints.

assembly: A grouping of one or more design components.

subassembly: An assembly placed in a larger assembly.

components: Parts and subassemblies used to form an assembly.

Assembly Model Fundamentals

Assembly files carry the extension .iam. One option for developing an assembly is to insert existing components into an assembly file and then assemble the components with constraints. This is an example of a process that some designers refer to as *bottom-up design*. This technique is appropriate if all or most components already exist. Depending on your approach and the complexity of the assembly, you can insert all components before applying constraints, as shown in **Figure 17-1**. A common alternative is to insert and constrain one or two components at a time.

Another option is to create new components in place within an assembly file. This is an example of a process that some designers refer to as *top-down design*. Both assembly techniques are effective, and a combination of methods is common. For some applications, however, it is faster, easier, and more productive to develop components in place. Developing components in an assembly file creates an assembly and a separate part or assembly file for each component. Chapter 18 describes building assemblies in place.

bottom-up design: A design approach that brings individual components together to form an assembly.

top-down design: A design approach in which the assembly controls, or produces, individual components.

Figure 17-1.
An example of guitar components, including parts and subassemblies, brought together to
form an assembly model. (Model courtesy of Ethan Collins)
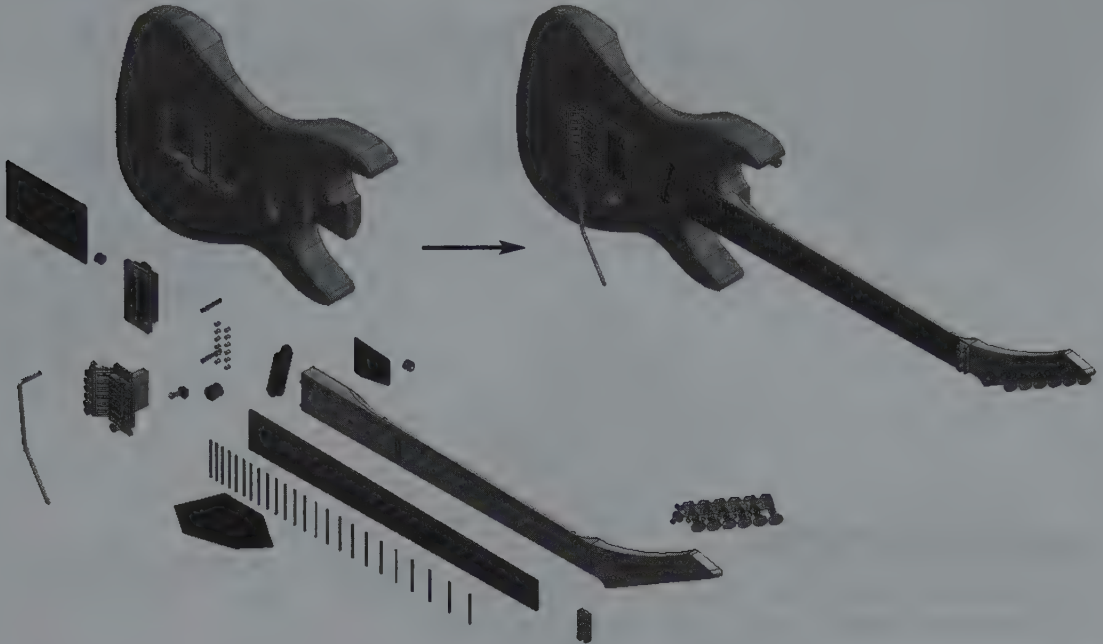
CAUTION

The **Assembly** environment includes molding tools that allow you to create sketches and some features, such as extrusions and holes. Use these tools only for post-assembly or similar processes, such as drilling a hole through assembled components. Do not confuse these feature tools with those used to create part or subassembly models, even though they function the same. Using feature tools to modify an assembly does not affect component models outside of the assembly file.

Exercise 17-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-1.

Exercise 17-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-2.

Inserting Components

The **Place Component** tool allows you to insert components using the **Place Component** dialog box, which is similar to opening a file with the **Open** dialog box. The **Files of type** drop-down list contains part, assembly, and component file options. Select **Component Files** to display part and assembly files. You can also choose to display components created using other CADD systems or those saved as STEP or SAT files. Once you locate components to insert, you can select multiple component files by pressing [Ctrl] or [Shift], or by using a crossing window. This allows you to insert several components without reusing the **Place Component** tool. The first component listed in the **File name** box is inserted first.

> **NOTE**
>
> Imported component files such as STEP or SAT files are not parametrically controlled. The buttons in the **iMates** area control the use of iMates, described later in this textbook.

The first component you insert, which is the first component listed in the browser, is typically a major part or subassembly, such as the assembly body or a main support base. The first assembly component is automatically *grounded* to the assembly origin, or center point, located in the **Origin** folder of the browser. See **Figure 17-2**. You can place additional ungrounded copies of the same component by picking locations, or by pressing [Enter]. To place a single component, or after you place multiple copies of a component, press [Esc] or right-click and select **Done**. Components you insert after the first are not grounded or constrained unless they include iMates.

grounded component: An assembly component fixed in position, with no freedom of movement.

> **NOTE**
>
> Toggle grounding of any component by right-clicking on the component in the graphics window or browser, and selecting or deselecting **Grounded**.

If the **Resolve Link** dialog box appears when you try to insert an existing component, the component file probably is not located in the current project, is not available because of a network problem (if contained in a local network), or has been changed.

Figure 17-2.
An example of an initial, grounded assembly component.

Visible center point

Visible default work planes
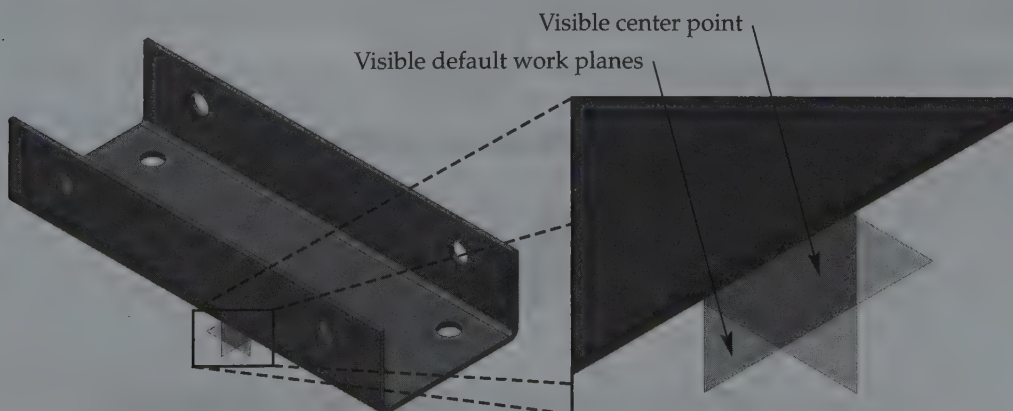
To ensure that you do not receive the **Resolve Link** dialog box, try to identify and solve the problem before opening the assembly. However, you can also pick the **Open** button in the **Resolve Link** dialog box and attempt to locate the file elsewhere, or skip the entire process by picking the **Skip** button.

> **NOTE**
>
> When saving an assembly, Inventor may ask you to save the assembly file and its *dependents*. Save dependents in addition to the assembly to ensure parametric relationships.
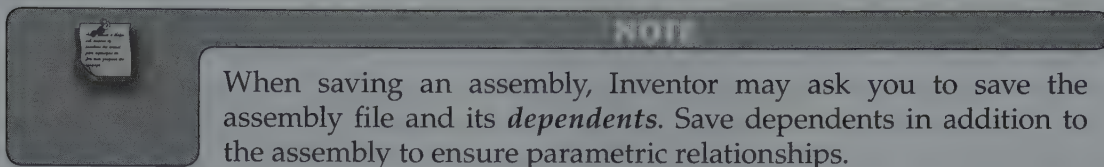
Exercise 17-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-3.

Alternative Insertion Methods

Inventor provides other methods of placing components. One option is to open the assembly file that will receive components and the component files to insert. Arrange all windows so they are visible as shown in **Figure 17-3**. Next, select a component window to activate it, and then drag and drop the component from the *browser* into the assembly window. You can also drag and drop multiple components from an assembly file into a different assembly file.

Another technique is to open the assembly file that will receive components, and then access Microsoft Windows Explorer and locate the component files to insert. Drag and drop the components from Explorer into the assembly graphics window. If the Inventor window is not visible, drag the components to the Inventor button on the taskbar and hold them there until the Inventor window appears. Then drop the components into the assembly graphics window.

Figure 17-3.
Inserting a component (**Engine Head.ipt**) into an assembly (**Engine MKII.iam**) by dragging the component from the browser into the assembly window.

Drag the component name to the assembly window

Active window
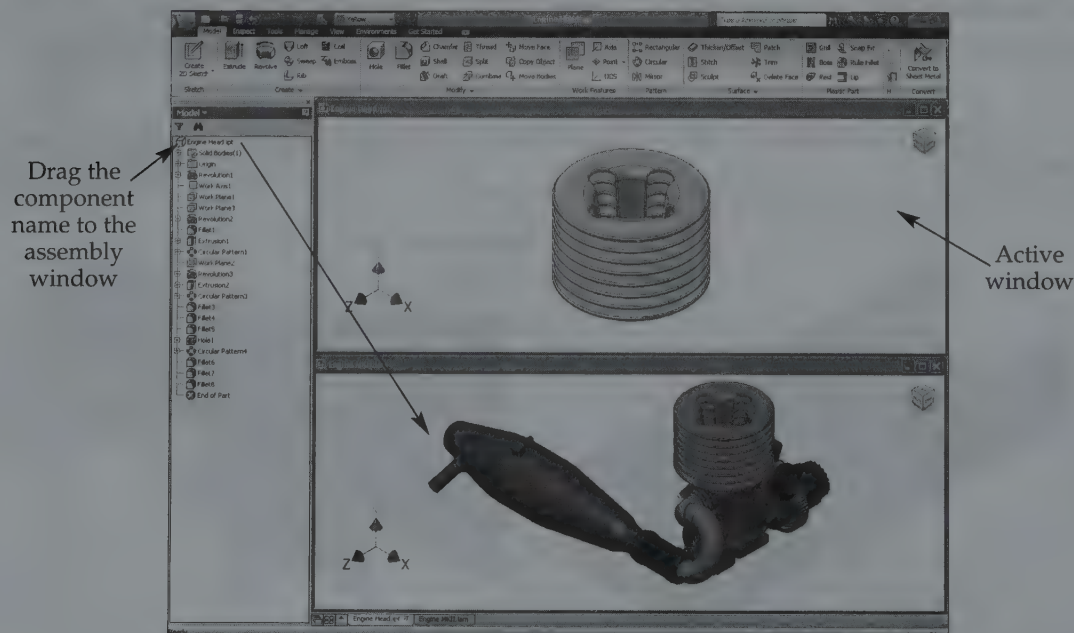
i-drop: The process of dragging and dropping shared content into component files; also the tool used for this process.

third-party (shared) content: Files available on the Internet, such as bolts from a bolt manufacturer, or components accessible on an intranet system, such as standard parts used for developing assemblies.

Exercise 17-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-4.

Constraining Assemblies

Once you insert or create assembly components, the typical next step is to add constraints. Constraining an assembly replicates the process of assembling a product. Each constraint removes a certain amount of movement freedom. You can *drive* remaining freedom of movement to examine design concepts. Some constraints automatically occur when you create components in place, which is an advantage to in-place development. However, even components built in place may require additional constraints or constraint adjustments.

drive: Manipulate to see the amount of movement between components, pause movement, see adaptivity, and detect collisions between components

> **NOTE**
>
> You can observe assembly degrees of freedom by dragging components or using the **Degrees of Freedom** tool.

Ribbon
View
> Visibility

Degrees of Freedom

Component geometry and design requirements determine the required constraints. Grounding is a type of assembly constraint. You can ground any component, but you typically only ground the base component and components that do not move in reference to other components. In addition to grounding, you can apply *assembly constraints, motion constraints*, and *transitional constraints* as needed, depending on the application.
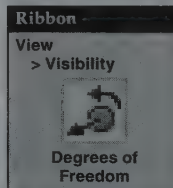
assembly constraints: Constraints that establish geometric relationships and positions between one component face, edge, or axis and another component face, edge, or axis.

motion constraints: Constraints that define the desired movement between one component and another using a specified ratio and direction.

Place Constraint Tool

The **Constrain** tool allows you to assemble components using the **Place Constraint** dialog box. See **Figure 17-4**. Select the tab that corresponds to the type of constraint to create. Then use the available options and select geometry on each component. Depending on the constraint, you may be able to select points, edges, axes, faces, or planes. Often it is necessary to select a combination of items, such as a face and edge.

transitional constraints: Constraints that identify relationships between the transitioning path of a fixed component and a component moving along the path.

Figure 17-4.

The **Assembly** tab of the **Place Constraint** dialog box with the **Mate** constraint button selected, and a basic example of a mate solution and two flush solution mate constraints between two parts.

Mate constraint between the top of one feature face and the bottom of another

Flush constraint between two faces

Angle

Tangent

Insert

Mate

Mate button

Flush button

Flush constraint between two faces
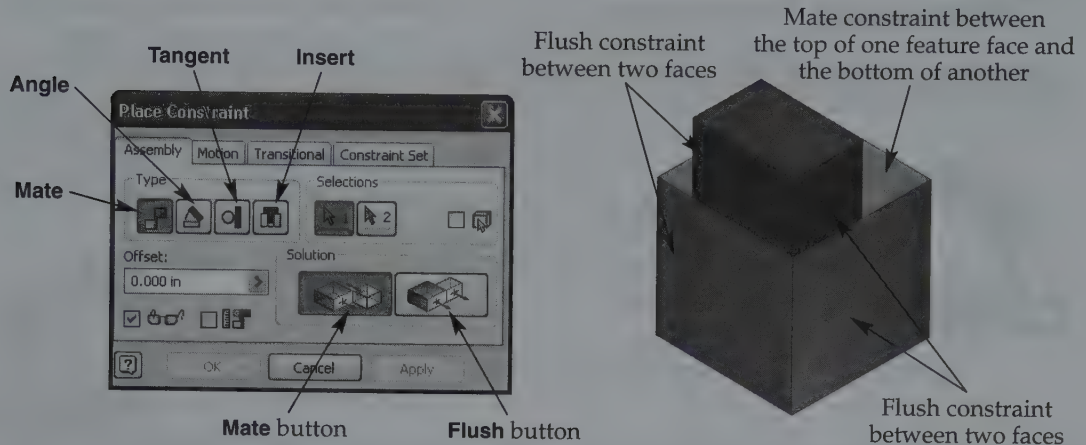
NOTE

When applying constraints, always remember that the first component you select moves to constrain to the second component you select, which does not move. However, if existing constraints prohibit this action, the second selection moves to the first.

Assembly Constraints

Most assemblies include several assembly constraints specified using options in the **Assembly** tab. Typically, you place assembly constraints before you add motion or transitional constraints. Choose an assembly constraint type button to form the corresponding constraint.

Mate

Select the **Mate** button to establish a mate or flush constraint between component faces, planes, axes, edges, or points. Refer again to **Figure 17-4**. Select a button in the **Solution** area to begin the process, although you can change the solution at any time before applying the constraint. Pick the **Mate** button to create a *mate solution*, as shown in **Figure 17-5**. Select the **Flush** button to form a *flush solution* as shown in **Figure 17-6**.

Next, select component geometry to constrain. When you are working on a complex or detailed assembly, it may help to check **Pick Part First**, which allows you to pick a part to isolate it from other assembly components and makes it easier to choose geometry on the selected part. Use the **First Selection** and **Second Selection** buttons to pick geometry on mating components. **Figure 17-7** shows examples of common selections. Remember, if the **Pick Part First** check box is active, you must pick components before you can make selections. Then specify the offset between components using the **Offset** text box. Use an offset of 0 to form a contacting fit.

Another initial approach is to select the **Predict Offset and Orientation** check box. This allows Inventor to predict how components should constrain based on the location and selection of components. If the arrows that appear when you select geometry point in the same direction, Inventor predicts a flush constraint. If the arrows point in opposite directions, Inventor predicts a mate constraint. Inventor assigns an offset value based on the current offset of the selected components.

The **Show Preview** check box is active by default, allowing you to preview the effects of the constraint. If the constraint looks correct, pick the **Apply** button to form the constraint, or pick the **OK** button to apply the constraint and exit the tool.

Ribbon

Assemble > Position

Constrain

Type

[C]
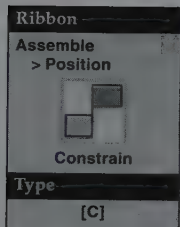
mate solution: A constraint that places two faces along the same plane facing in opposite directions, two axes collinear to each other, two edges collinear to each other, or two points matched together.

flush solution: A constraint that positions two faces along the same plane, facing the same direction.

Figure 17-5.
A mate constraint with a mate solution applied to planar faces, axes, edges, and points.

Two faces along the same plane, facing in opposite directions

Two axes collinear to each other

Two edges collinear to each other

Two points matched together
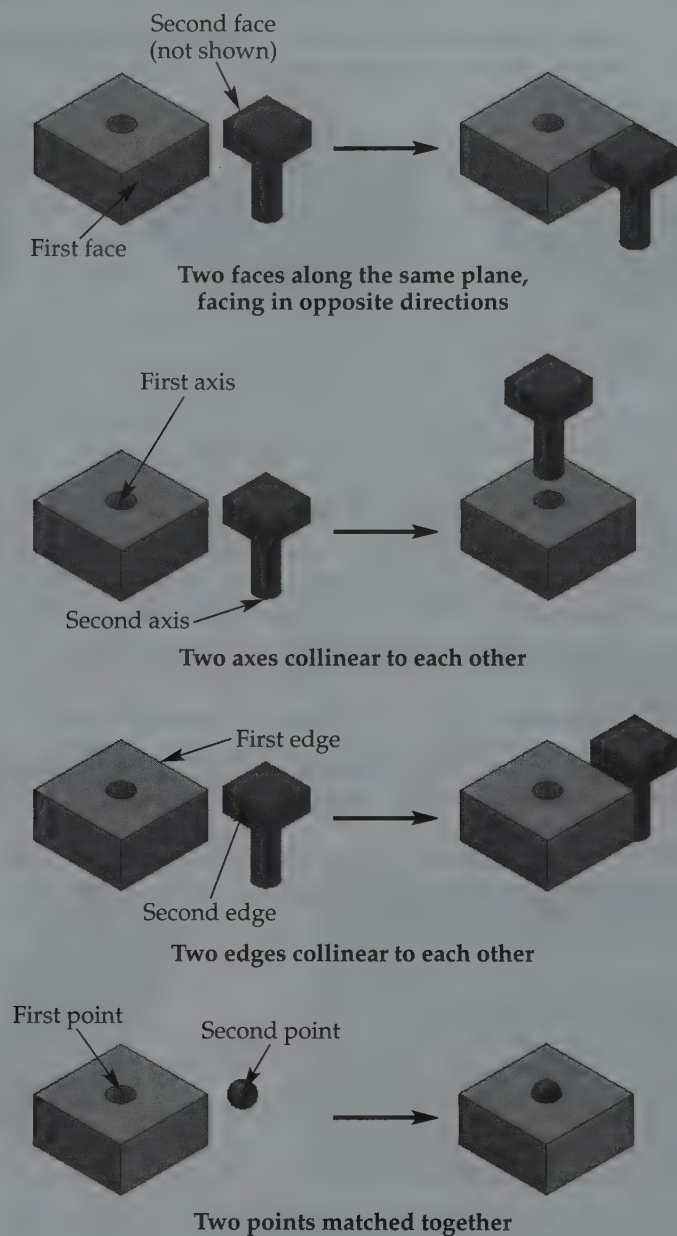
Figure 17-6.
A mate constraint with a flush solution places the selected faces in the same plane and facing the same direction.
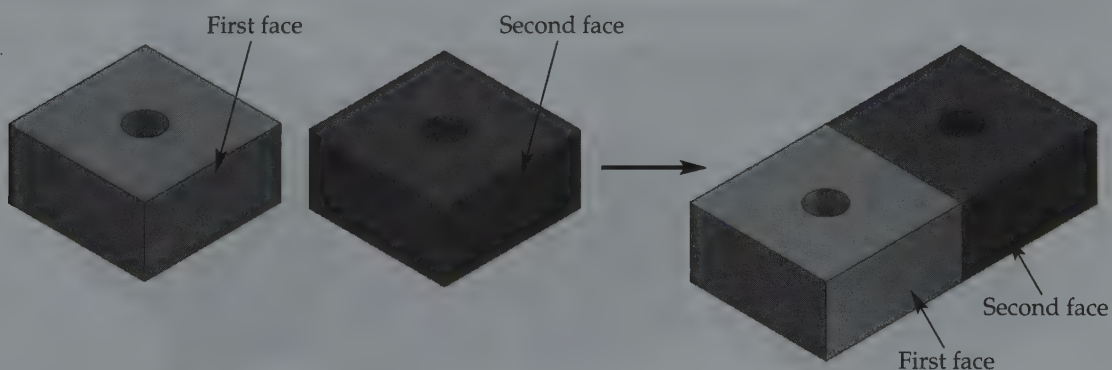
Figure 17-7.
Select one of these component elements after selecting the component.

Selecting a
Component Face

Selecting a
Component Edge

Selecting a
Component Axis

Selecting a
Component Point
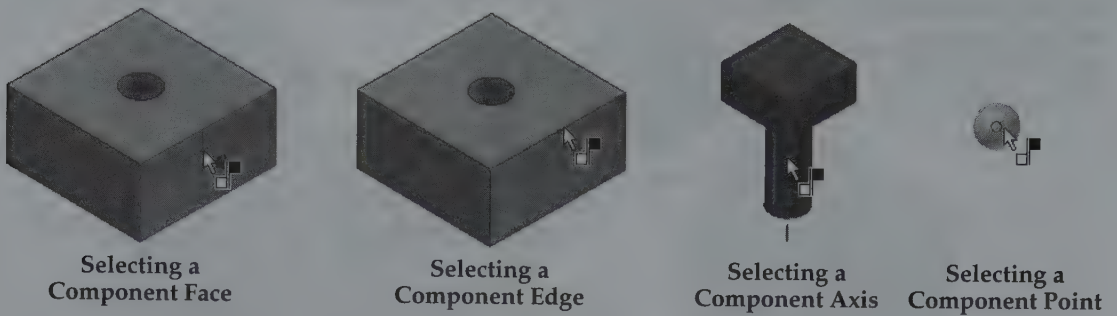
Angle

Select the **Angle** button to establish an angle constraint between component faces, axes, or edges. See **Figure 17-8**. Use the **First Selection** and **Second Selection** buttons to pick component geometry. If necessary, use the **Pick Part First** check box to aid selection. **Figure 17-9** shows how applying the same 0° angle constrains the assembly

Figure 17-8.
The **Assembly** tab of the **Place Constraint** dialog box with the **Angle** constraint button selected, and an example of a 45° angle constraint between two selected faces.

Angle constraint button

First selected face

Second selected face
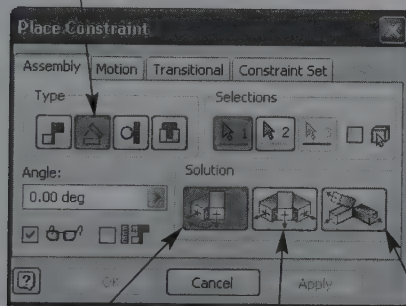
Directed Angle button

Undirected Angle button

Explicit Reference vector button
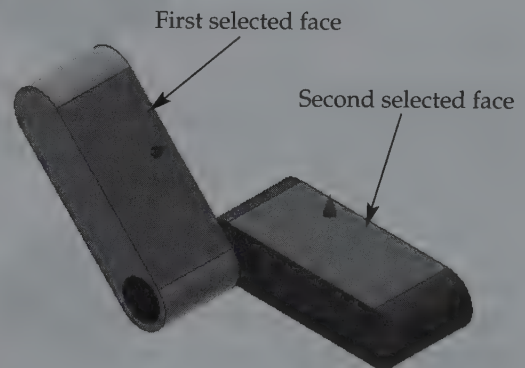
Figure 17-9.
A—A 0° angle constraint between two selected edges. B—A 0° angle constraint between an edge and face. C—A 0° angle constraint between an edge and axis.

Selected edges

Selected edge

Selected edge

Selected axis

Selected face

A

B

C
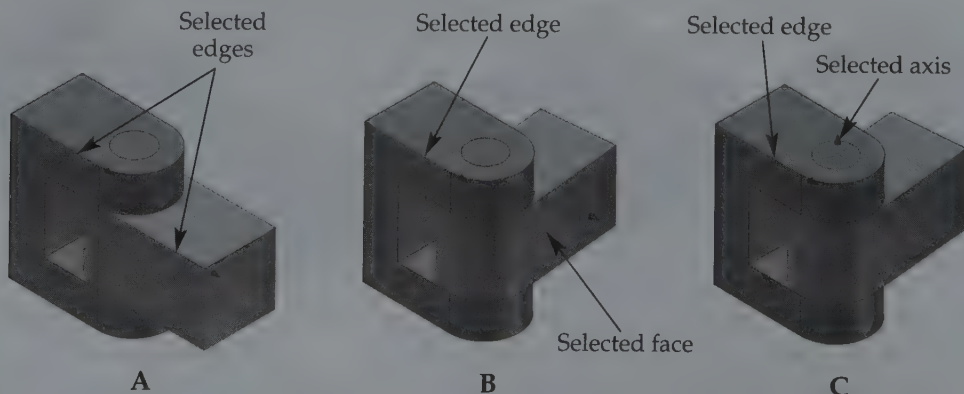
differently depending on selected geometry. Specify the angle between components using the **Angle** text box.

Another initial approach is to select the **Predict Offset and Orientation** check box. This allows Inventor to predict how components should constrain based on the location and selection of components. For example, if the arrows that appear when you select geometry point in opposite directions, Inventor predicts a 180° angle.

The button that you select in the **Solution** area mainly affects how components interact when they move. The default **Directed Angle** button is appropriate for basic applications, and allows you to enter a positive or negative angle from 0°. When you move or drive components, the angle is always measured from the same direction, such as clockwise.

In some cases, when you move or drive angle constraints, components flip unexpectedly into an unacceptable orientation. You may be able to solve this problem by picking the **Undirected Angle** button, which allows the angle to be measured from either direction. If components still flip incorrectly, pick the **Explicit Reference Vector** button. The **Third Selection** button becomes enabled, allowing you to choose a face, plane, edge, or axis as a vector direction. See **Figure 17-10**. The selected vector fixes the position to eliminate possible flipping of components during movement.

PROFESSIONAL TIP

The **Explicit Reference Vector** option is especially useful when components are not fully constrained, or when a base component is not grounded, as shown in **Figure 17-10**.

The **Show Preview** check box is active by default, allowing you to preview the effects of the constraint. If the constraint looks correct, pick the **Apply** button to form the constraint, or pick the **OK** button to apply the constraint and exit the tool.

Tangent

Pick the **Tangent** button to define a tangent relationship between a planar face, curved face, or edge of one component and the curved face or plane of another component. See **Figure 17-11**. Select a button in the **Solution** area to begin the process,

---

Figure 17-10.
An example of when the **Explicit Reference Vector** solution may be necessary. The solution allows for fluid movement between components when you analyze freedom of movement and drive constraints.

Second selected
reference plane

First selected
reference (plane)

Enabled when the
Explicit Reference Angle
button is selected

Selected explicit
reference vector (plane)
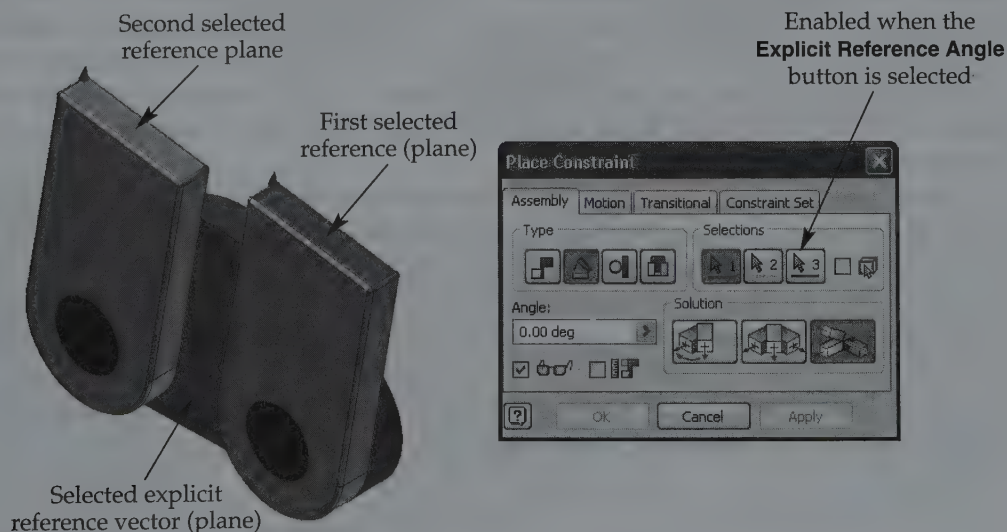
Figure 17-11.
The **Assembly** tab of the **Place Constraint** dialog box with the **Tangent** constraint button selected, and examples of inside and outside tangent constraints. Notice how one of the two selections must be a tangent face.

Select first
Select second

Select first

Select second   Selected tangent faces

**Inside Tangent
Constraint Solution**

**Outside Tangent
Constraint Solution**

**Outside Tangent
Constraint Solution**
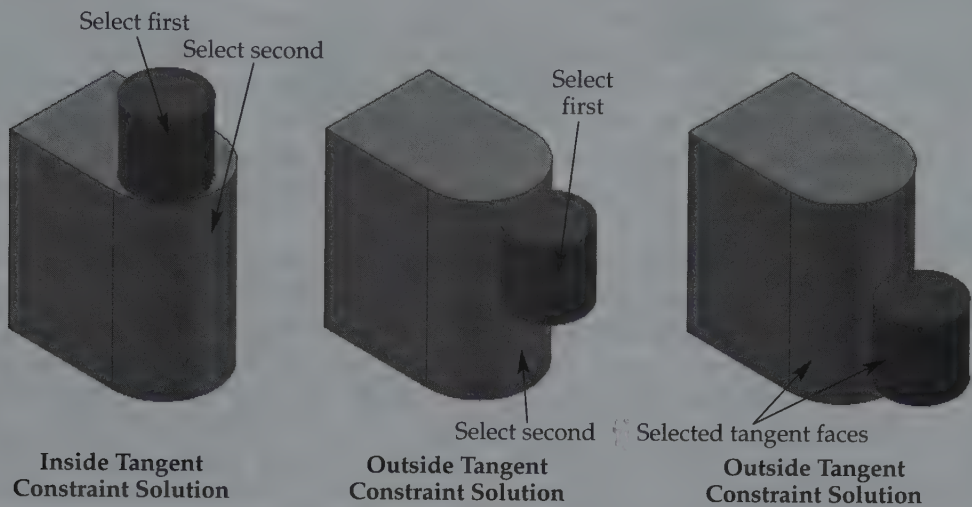
although you can change the solution at any time before applying the constraint. Pick the **Inside** button to place the component you select first inside the component you select second. Pick the **Outside** button to position the component you select first outside of the component you select second.

Next, use the **First Selection** and **Second Selection** buttons to pick component geometry. If necessary, use the **Pick Part First** check box to aid selection. As shown in **Figure 17-11**, one of the two selections must be a tangent face. Then specify the offset between components using the **Offset** text box. Use an offset of 0 to form a contacting fit. The **Show Preview** check box is active by default, allowing you to preview the effects of the constraint. If the constraint looks correct, pick the **Apply** button to form the constraint, or pick the **OK** button to apply the constraint and exit the tool.

Insert

Select the **Insert** button to link two components that require an axis-to-axis and edge-to-edge mate constraint, such as the constraints typically found between a hole and a screw or shaft, or a tube and a cap. **Figure 17-12** shows an insert constraint between the axis of a bushing and the axis of a housing hole, and between the inside edge of the bushing and the inside edge of the housing hole. Select a button in the **Solution** area to begin the process, although you can change the solution at any time before applying the constraint. Pick the **Opposed** button to assemble components as if using a mate constraint, mate solution. Pick the **Aligned** button to assemble components as if using a mate constraint, flush solution. **Figure 17-13** shows examples of using the **Opposed** and **Aligned** solutions.

Next, use the **First Selection** and **Second Selection** buttons to pick component edges. If necessary, use the **Pick Part First** check box to aid selection. Selecting an edge picks the edge and an axis. You must select corresponding edges on both components to ensure the proper constraints. Specify the offset between components using the **Offset** text box. Use an offset of 0 to form a contacting fit. The **Show Preview** check box is active by default, allowing you to preview the effects of the constraint. If the constraint looks correct, pick the **Apply** button to form the constraint, or pick the **OK** button to apply the constraint and exit the tool.

Figure 17-12.
The **Assembly** tab of the **Place Constraint** dialog box with the **Insert** constraint button selected, and an example of using the **Insert** constraint to center and mate a bushing in a housing. Use the **Insert** constraint for most shaft-in-hole types of assembly applications.

Insert constraint button   Pick Part First check box

Opposed button   Aligned button

Selected component edge and axis

Selected component edge and axis

Selected component edge and axis

Constrained components

Bushing   Housing   Completed Assembly
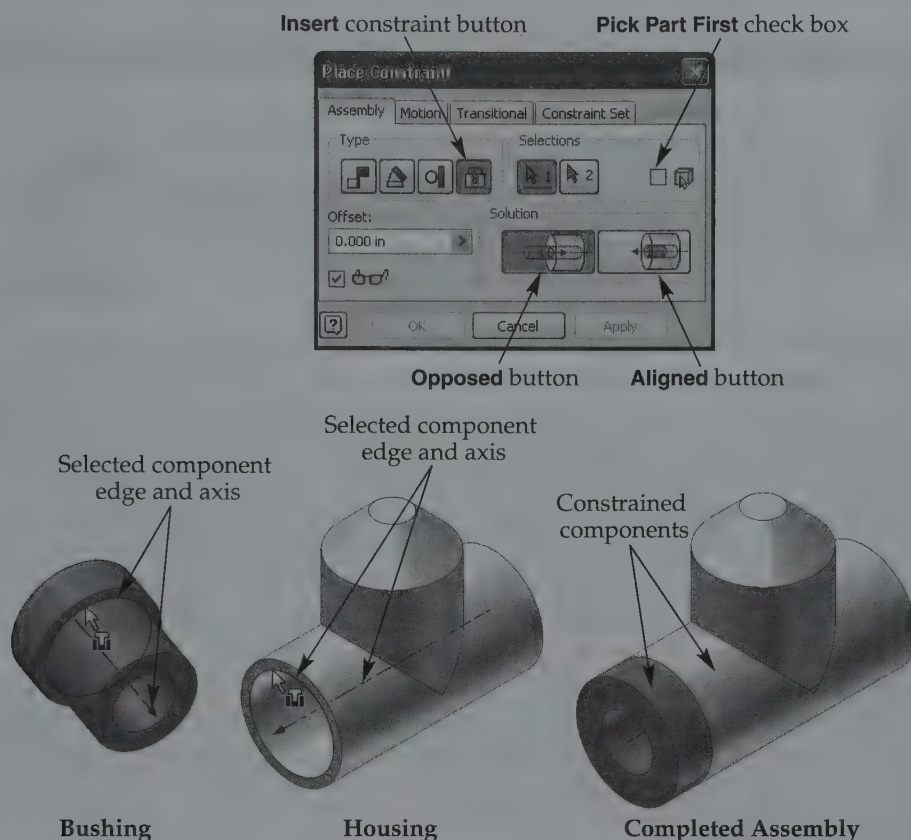
Figure 17-13.
Selecting an **Aligned** or **Opposed** solution determines the direction of the insert.

Opposed Solution   Aligned Solution
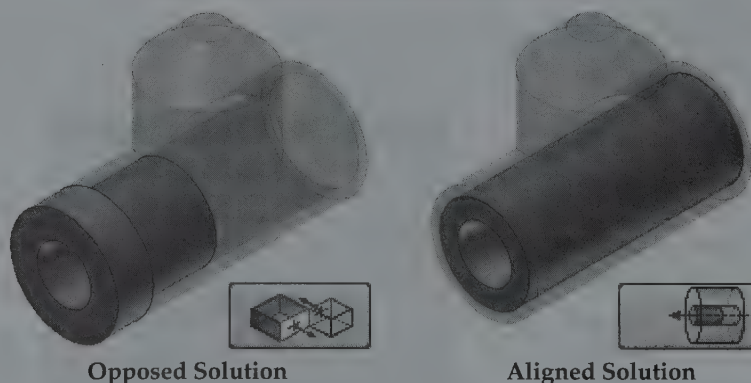
Exercise 17-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-5.

Motion Constraints

The **Motion** tab, shown in **Figure 17-14**, provides options for defining movement between components according to ratio and direction. Only components that rotate and move together, such as gears, pulleys, racks and pinions, and bearings, require

motion constraints. For example, a motion constraint can identify whether two gears move in the same direction or opposite directions, and it can specify the ratio of movement between one component and another. Motion constraints do not locate, position, or fix components.

> **NOTE**
>
> Typically, you add assembly constraints before motion constraints to establish geometric relationships, such as flush gear faces and mates between holes and shafts. It is often necessary to suppress and/or drive certain constraints in order to observe a motion constraint.

Rotation

Select the **Rotation** button to establish a rotation motion constraint, which controls the direction of rotation and the ratio of movement between two revolving components such as gears, pulleys, and wheels. See **Figure 17-14**. Use the **First Selection** and **Second Selection** buttons to pick component geometry. If necessary, use the **Pick Part First** check box to aid selection. **Figure 17-14** shows examples of selections that create the same constraint between two gears. Selecting component elements is not as critical when applying motion constraints, because motion constraints do not define component position. However, the selection pattern does specify the ratio. Therefore, if you define a 3:1 ratio, the first component you select should be the component that revolves once for every three times the second component rotates.

The default **Forward** solution button creates a forward motion that allows one component to rotate in the same direction as the other component. Pick the **Reverse** solution button to reverse the rotation. A reverse motion allows one component to rotate in the opposite direction of the other component. The arrows shown in **Figure 17-14** identify the rotation direction.

Figure 17-14.
The **Motion** tab of the **Place Constraint** dialog box with the **Rotation** constraint button selected, and selection options for establishing a rotational constraint between gears.

Selecting Two Planar Faces   Selecting Two Axes   Selecting Two Edges
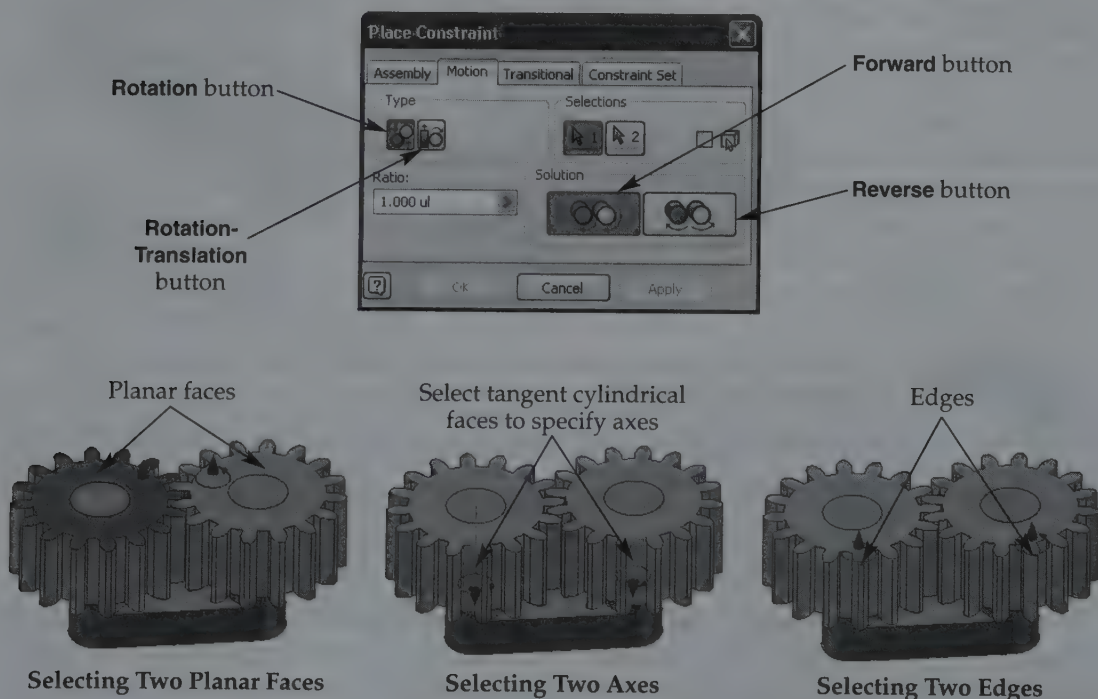
Specify a rotation ratio using the **Ratio** text box. For example, if you apply a rotation constraint between the same size gears, as shown in **Figure 17-14,** enter 1 in the **Ratio** text box to correspond to a 1:1 ratio. If the first gear, wheel, or pulley you select is half the size of the second component and rotates twice for every single rotation of the second component, enter .5 in the **Ratio** text box to correspond to a 1:2 ratio. Pick the **Apply** button to add the constraint, or pick the **OK** button to add the constraint and exit the tool.

Exercise 17-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-6.

Rotation-Translation

Pick the **Rotation-Translation** button to establish a direction of rotation and a rotation distance between a rotating component and a translating component, such as a rack and pinion, a wheel and part, or a screw and hole thread. See **Figure 17-15.** Use the **First Selection** and **Second Selection** buttons to pick component geometry. If necessary, use the **Pick Part First** check box to aid selection. Selection options include edges, axes, planar faces, and tangent faces. **Figure 17-15B** shows appropriate selections for a rack and pinion.

The default **Forward** solution button creates a forward motion that allows the rotating component to rotate in the same direction as the sliding component. Pick the **Reverse** solution button to change the motion to a reverse rotation. A reverse motion allows the rotating component to rotate in the opposite direction of the sliding component. The arrows shown in **Figure 17-15** identify the rotation direction.

Specify a rotation-translation distance between the selected components using the **Distance** text box. The distance controls how far the sliding component (second selection) moves in reference to the rotating component (first selection). For example, if you use a distance of 3″, for every 360° the first component rotates, the second component moves 3″. A distance of 0 does not allow the second component to move in reference to the revolution of the first component. Pick the **Apply** button to add the constraint, or pick the **OK** button to add the constraint and exit the tool.

Figure 17-15.
A—The **Motion** tab of the **Place Constraint** dialog box with the **Rotation-Translation** constraint button selected. B—An example of selecting a planar face and an edge to add rotation-translation constraint.
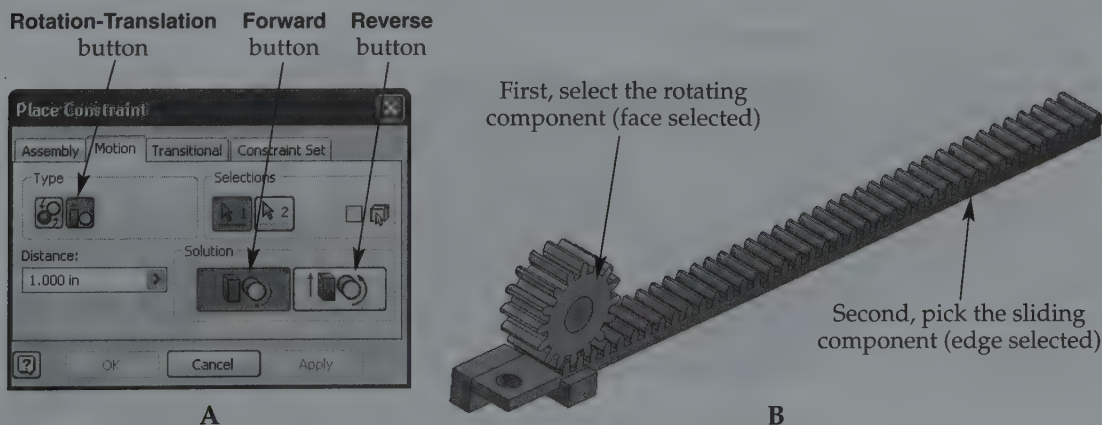
Exercise 17-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-7.

Transitional Constraints

The **Transitional** tab, shown in **Figure 17-16**, provides options for defining movement between a component and a path, such as a roller and a cam. A transitional constraint links faces along a path, applying constant contact between two planar or tangent components. You must use assembly constraints to establish all other constraints, such as a flush relationship between the planar roller face and the cam face shown in **Figure 17-16**.

> **NOTE**
>
> Typically, you add assembly constraints before transitional constraints. It is often necessary to suppress and/or drive certain constraints in order to observe a transitional constraint.

Use the **First Selection** and **Second Selection** buttons to pick planar or tangent faces. If necessary, use the **Pick Part First** check box to aid selection. The **Show Preview** check box is active by default, allowing you to preview the effects of the constraint. If the constraint looks correct, pick the **Apply** button to form the constraint, or pick the **OK** button to apply the constraint and exit the tool.

> **PROFESSIONAL TIP**
>
> Help locate geometry to constrain using the **Select Other** tool and wheel mouse selection techniques. You can cycle through possible edges, axes, planes, and other elements for constraint purposes.

Exercise 17-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-8.

Figure 17-16.
The **Transitional** tab of the **Place Constraint** dialog box, and an example of a transitional constraint application in which the roller moves along the faces of the cam.
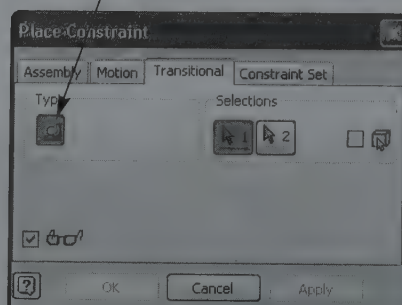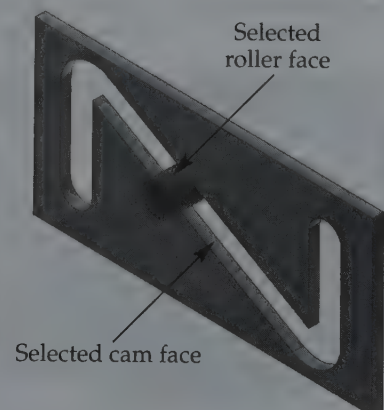
Constraint Sets

Pick the **Constraint Sets** tab to mate components according to UCSs. Use the **First Selection** and **Second Selection** buttons to pick an appropriate UCS on each component. If necessary, use the **Pick Part First** check box to aid selection. The **Show Preview** check box is active by default, allowing you to preview the effects of the constraint. If the constraint looks correct, pick the **Apply** button to form the constraint, or pick the **OK** button to apply the constraint and exit the tool.

Alt-Drag

Alt-drag is an alternative assembly constraint function that does not require using the **Constrain** tool. Alt-drag is similar to moving an unconstrained component to a new location by dragging, but before picking the component to move, hold down [Alt]. Then hold down the left mouse button on a component point, edge, axis, face, or plane to constrain, and drag the component to the corresponding location on the other component. As you drag the component over other components, a preview of constraint options appears. See **Figure 17-17**.

If the incorrect constraint appears, you may be able to establish the correct constraint by moving the component slightly, or you can toggle between different constraints using hot keys. To use hot keys, while holding down the left mouse button, release [Alt] and press one of the keys shown in **Figure 17-18** to activate the specified constraint and solution. If the constraint type is correct, but the solution is incorrect, press the space bar to toggle solutions. You may still have to drag the component to preview the constraint. When you are satisfied with the intended constraint, release the left mouse button to apply the constraint. View the browser to confirm the correct constraint.

alt-drag: Establishing assembly constraints, including mate, flush, tangent, and insert constraints, by dragging one component to another component.

PROFESSIONAL TIP

Use the **Select Other** tool and wheel mouse selection techniques during the drag-mate process to cycle through possible edges, axes, planes, and other elements for constraint purposes.

Figure 17-17.
Hold down [Alt] and drag the face, edge, or point of the moving component to the corresponding location on the stationary component.

Selected edge

Insert icon appears during the move

Stationary component

Moving component

Drag the selected edge of the moving component to this edge

Selected edge is highlighted during drag-mate constraining

Constrained assembly
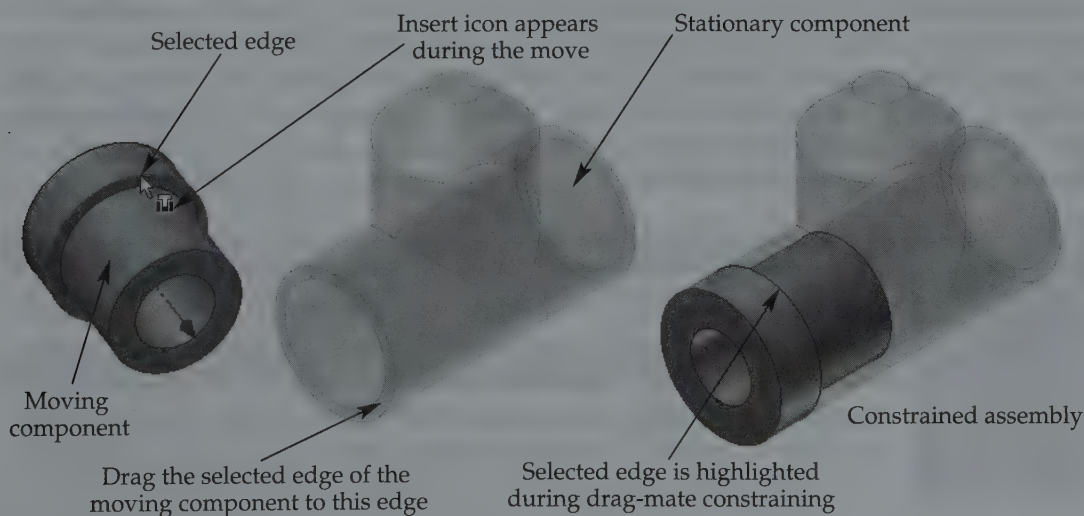
Figure 17-18.
Options for changing drag-mate constraints. Press the space bar to pick an alternative solution.

Keys	Function *(Select either key to change the current constraint to a/an:)*
[A] or [2]	Angle constraint.
[I] or [4]	Insert constraint, opposed direction.
[M] or [1]	Mate constraint, flush solution.
[R] or [5]	Rotation motion constraint. Press the space bar to change the rotation direction, if needed.
[S] or [6]	Translation motion constraint. Press the space bar to change the slide direction, if needed.
[T] or [3]	Tangent constraint. Press the space bar to change to an inside or outside tangent constraint.
[X] or [8]	Transitional constraint.

iMates: Constraints added to individual components for later assembly.

Supplemental Material

iMates

For information about constraining using *iMates*, go to the Student Web site (www.g-wlearning.com/CAD), select this chapter, and select **iMates**.

Editing Constraints

Expand a component in the browser to view constraints. To temporarily disable the effects of a constraint without deleting the constraint, right-click on the constraint and select **Suppress**. Suppressing a constraint is often required to explore design options or drive constraints. Right-click on a constraint and select **Edit** to make changes to the constraint using the **Edit Constraint** dialog box. The **Edit Constraint** dialog box offers the same options as the **Place Constraint** dialog box.

The **Value** text box provides an easier way to edit a constraint value, such as offset, angle, ratio, or distance, than using the **Edit Constraint** dialog box. Pick a constraint in the browser to access the **Value** text box. See **Figure 17-19A**. Then specify a new value and press [Enter]. A second option is to double-click on a constraint in the browser, or right-click on a constraint in the browser and select **Modify**, to access the **Edit Dimension** dialog box. See **Figure 17-19B**. Enter a new value in the text box and pick the **OK** button. A third option is to use the **Parameters** dialog box shown in **Figure 17-19C**.

Ribbon

Manage > Parameters

fx

Parameters
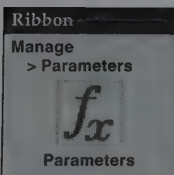
PROFESSIONAL TIP

A constraint is divided between two components. As a result, the same constraint appears with two components in the browser. To locate and highlight the corresponding half of a constraint, right-click on a constraint in the browser and select **Other Half**. When you move the cursor over a constraint in the browser, an extensive tooltip appears. Use these tooltips to help locate and clarify constraint relationships.

Figure 17-19.

Figure 17-19.
A—Adjusting a constraint value using the **Value** text box. B—Adjusting a constraint value using the **Edit Dimension** dialog box. C—Access the **Parameters** dialog box in the assembly environment to view and control assembly constraints and assembly model parameters.

Value edit box

A

B

Parameter Name	Unit	Equation	Nominal Value	Tol	Model Value	Export Parameter	Comment
Model Parameters							
d2	in	0.000 in	0.000000	○	0.000000	☐	
d3	in	0.000 in	0.000000	○	0.000000	☐	
d4	in	0.000 in	0.000000	○	0.000000	☐	
d5	in	0.000 in	0.000000	○	0.000000	☐	
d12	in	0.000 in	0.000000	○	0.000000	☐	
d13	deg	0 deg	0.000000	○	0.000000	☐	
d14	in	0.000 in	0.000000	○	0.000000	☐	
d15	deg	45.00 deg	45.000000	○	45.000000	☐	
d16	deg	360.00 deg	360.000000	○	360.0000...	☐	
d17	deg	1.00 deg	1.000000	○	1.000000	☐	
d18	ul	1.000 ul	1.000000	○	1.000000	☐	

☐ Display only parameters used in equations   Reset Tolerance

Add   Link

Done

C

Exercise 17-9

Driving Constraints

Dragging components is an effective way to analyze design progress and view the effects of constraints. Hold down the left mouse button on a component and rotate, slide, or offset the component, depending on the design and existing constraints. Driving constraints is another valuable function, especially for analyzing components that are difficult to drag and to view specific movement characteristics.

Assuming an acceptable driving constraint is available, typically an angle or mate, drive the constraint by right-clicking on the constraint in the browser and selecting **Drive Constraint**. The **Drive Constraint** dialog box appears with the appropriate angular or linear units for the selected constraint. **Figure 17-20** shows an example of driving a mate-angle constraint. In this example, –112° is the drive start angle specified in the **Start** text box, and 112° is the drive end angle specified in the **End** text box. The start and end values indicate the total acceptable component movement. In this example, the values specify the angles from 0° in one direction (–112°) and the other (112°) that the movable component can rotate before colliding with the grounded component.

NOTE

Specify start and end offset values when driving a mate constraint. Adjust the start and end values as needed.

Figure 17-20.
An example of using the **Drive Constraint** dialog box to drive a mate-angle constraint.
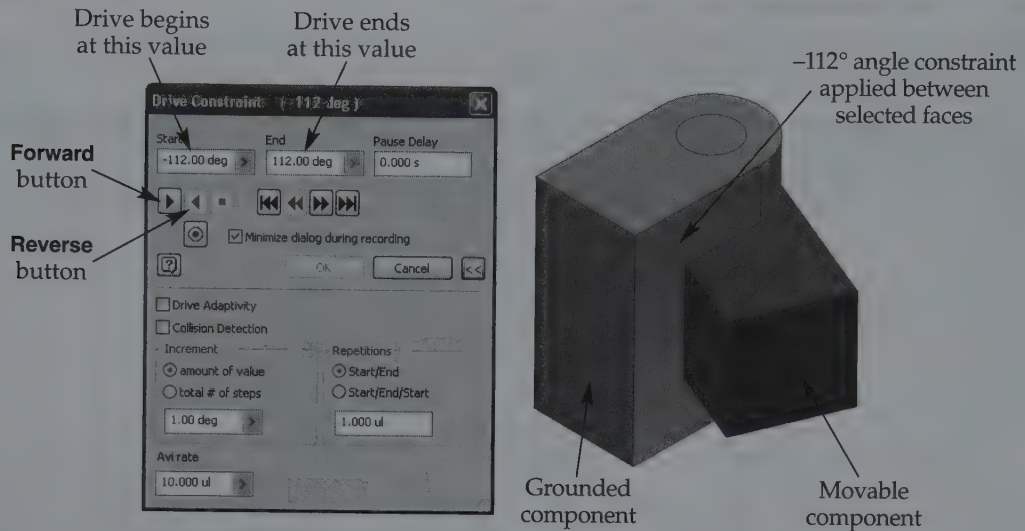
Driven constraints move a certain number of steps. For example, you can set an angle constraint to move 5° per step or a mate constraint to move .25″ per step. Use the **Pause Delay** text box to specify the number of seconds between steps. The default value is 0, which results in no pause between steps.

The **Forward** and **Reverse** buttons are available only when the **Start** and **End** text boxes have values and the constraint is driven to an appropriate location. Pick the **Forward** button to drive the constraint forward and the **Reverse** button to drive the constraint in reverse. If necessary, pick the **Pause** button while driving a constraint to stop the component movement. Use the **Minimum** button to return to the start position. Pick the **Reverse Step** button to drive the constraint a single step in the reverse direction, and the **Forward Step** button to drive the constraint a single step in the forward direction. Use the **Maximum** button to return to the end position.

Pick the **Drive Adaptivity** check box, with the **Collision Detection** check box deselected, to drive the adaptive properties of components in relation to the constraint. Select the **Collision Detection** check box, with the **Drive Adaptively** check box deselected, to receive an alert if collision occurs while driving. As shown in **Figure 17-21**, the components involved in the collision highlight, and the collision detection message appears. Pick the **OK** button to accept the collision and modify the drive, constraints, or component geometry.

Figure 17-21.
An example of detecting a collision while driving.
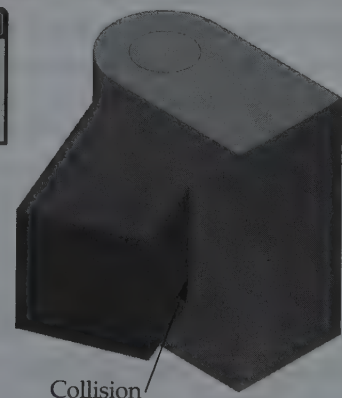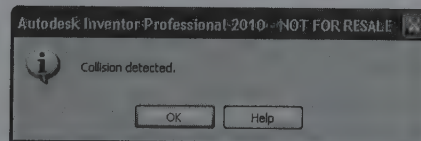
Pick the **Amount of value** radio button to define a certain increment value, such as 10° for a driven angle constraint, or 5″ for a driven mate constraint. Then specify the value using the **Increment** text box. A high increment value results in fewer increments and quicker movement. Pick the **Total # of steps** radio button to define a certain number of steps, such as 5 or 10. Using these examples, the total drive movement is divided into 5 or 10 steps. Use the **Increment** text box to specify the number of steps. A higher number of steps results in more increments and slower movement.

Pick the **Start/End** radio button to drive the constraint from the start position to the end position. Pick the **Start/End/Start** radio button to cause the drive to return to the start after it meets the end position. Then identify the number of times you want the drive to move from start to end, or from start to end to start, by specifying a value in the **Repetitions** text box.

Recording and Finalizing

Pick the **Record** button to record the drive as an animation file. The **Save As** dialog box appears, allowing you to specify a location for the animation file. You can also choose the file format by selecting AVI or WMV from the **Save as type:** drop-down list. An additional dialog box appears, allowing you to specify animation file parameters.

When you return to the **Drive Constraint** dialog box, pick the **Minimize dialog during recording** check box to minimize the **Drive Constraint** dialog box while recording the drive so that it does not appear in the animation. The **Avi rate** drop-down list is available when you save the animation file in an AVI format. The AVI rate defines the number of increments copied during the animation recording.

You can apply changes made to the constraint value while driving by picking the **OK** button. This also closes the **Drive Constraint** dialog box. To exit the tool without saving the modified value, press [Esc] or pick the **Cancel** button. A recorded animation is viable once you exit.

PROFESSIONAL TIP

While driving a constraint, use tools such as **Rotate**, **Pan**, **Zoom**, and **Look At** to analyze design movement.

Exercise 17-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 17-10.

Chapter Test

1. What is an assembly?
2. What is a subassembly?
3. What are components?
4. Briefly describe bottom-up design.
5. How does top-down design work?
6. Describe a grounded component.
7. Define *dependents*.
8. What are assembly constraints?
9. What are motion constraints?
10. Describe transitional constraints.
11. What is a mate solution?
12. What is a flush solution?
13. What does *alt-drag* mean?
14. How do you temporarily disable the effects of a constraint without deleting the constraint, and why would you want to do this?
15. Briefly describe two basic ways to view the effects of constraints on an assembly.

Problems

1. Ensure that the Exercises and Problems project is active. Access the **Open** dialog box and pick the **Samples** library to navigate to the **Samples** folder. Open the scissors.iam file from the following folder: Samples/Models/Assemblies/Scissors. Explore the constraints applied to each component, but do not attempt to make changes. Drive the Angle:1 constraint. Make an AVI recording named P17-1.avi of the default drive parameters using the **Forward** button. Close the assembly file without saving.

2. Ensure that the Exercises and Problems project is active. Access the **Open** dialog box and pick the **Samples** library to navigate to the **Samples** folder. Open the Tuner.iam file from the following folder: Samples/Models/Assemblies/Tuner. Explore the constraints applied to each component, but do not attempt to make changes. Drive the Angle:1 constraint. Make an AVI recording named P17-2.avi of the default drive parameters using the **Forward** button. Close the assembly file without saving.

3–6 Instructions:

- Insert existing components or prepare new components for insertion to create the following assemblies.
- Use appropriate sketching techniques and fully constrain sketch geometry.
- Add all necessary assembly constraints.
- Use your own judgment and approximate dimensions when necessary.
- Add as much information as possible to the **iProperties** dialog box for component and assembly files. Assign the specified material and color to parts.
- Follow the specific instructions for each problem to create the assemblies.

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

3. **Title:** PUSH PIN
   Units: Inch
   Template: Assembly-IN.iam
   Part Number: IAA-004
   Project: PUSH PIN
   Save as: P17-3.iam
   Specific Instructions: Insert the push pin head, P6-7.ipt. Create a pin as a separate part file using the Part-IN.ipt template and the instructions below. Insert the pin, P17-3.ipt, and use an assembly-insert constraint to insert the pin into the head. Use an assembly-mate constraint with a mate solution to mate the XY planes of the pin and head to create the final assembly shown. To build the pin:
   A. Sketch a circle with a center coincident to the projected center point on the XZ plane.
   B. Extrude the sketch .75″ in the positive direction and then chamfer the lower extrusion edge to create a point using a .15″ × .02″ chamfer.
   C. In the **iProperties** dialog box, specify a title of PIN and a part number of IAA-004-02, name the project PUSH PIN, and specify the Steel, High Strength Low Alloy material.
   D. Change the color to Chrome and save the part as P17-3.ipt.

(continued)

4. **Title:** BOTTLE ASSEMBLY
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-014
   Project: BOTTLE
   Save as: P17-4.iam
   Specific Instructions: Insert the bottle part P9-5.ipt and the bottle cap P10-8.ipt. Insert the bottle first to ground it. Constrain the assembly using an assembly-insert constraint as shown. Add a rotation-translation constraint appropriate for the thread pitch. Drive the insert constraint.
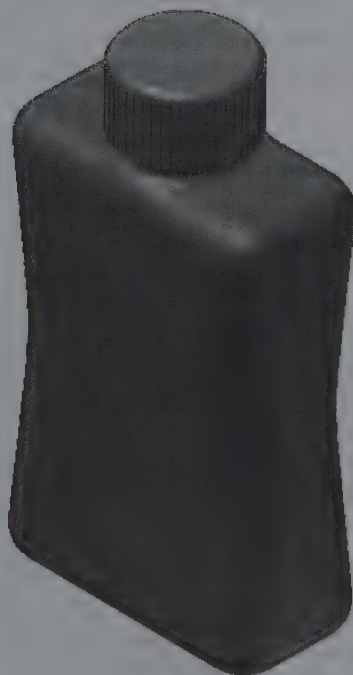
5. **Title:** PIPING
   Units: Metric
   Template: Assembly-mm.iam
   Part Number: IAA-020
   Project: PIPING
   Save as: P17-5.iam
   Specific Instructions:
   A. Insert one copy of P11-3.ipt (45° split elbow part) and two copies of P10-9 (45° elbow part) into the assembly file.
   B. Create a pipe with total length of 300 mm, as shown, in a separate part file. Give the pipe an inner diameter of 50 mm and an outer diameter of 56.8 mm. Use the dimensions of the elbow to size and locate the pipe flanges and holes. Name the file P17-5.ipt. In the **iProperties** dialog box, specify a title of PIPE and a part number of IAA-020-03, name the project PIPING, and specify the Steel, Mild material. Leave the color set to As Material.
   C. Add two copies of the pipe to the assembly.
   D. Constrain the assembly using appropriate assembly constraints as shown.

6. **Title:** WATCH BAND CLASP
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-103
   Project: WRIST WATCH
   Save as: P17-6.iam
   Specific Instructions: Design a watch band clasp similar to the example shown. Create five separate part files of the individual parts. Assign appropriate materials, colors, and properties to each component and save the parts using the file names P17-6a.ipt, P17-6b.ipt, P17-6c.ipt, P17-6d.ipt, and P17-6e.ipt. Insert the four pin parts into an assembly to create a subassembly (assembly file) named P17-6sub.iam. Assemble the remaining parts and the subassembly in the P17-6.iam file. The final assembly should look similar to the assembly shown.

Building Components in Place

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create components within an assembly file.
✓ Adapt components to design changes.

This chapter explains how to create components "in place" within an assembly file. Remember that when you create a component in place, the component forms in the assembly and is also saved as a separate component file. You will also learn to make models *adaptive*. Adaptive modeling is common when building components in place. It allows you to develop an assembly and analyze and preserve design intent as you model a product.

adaptive model:
A component that changes, or adapts, according to modifications made to an associated component.

Create Component Tool

Access the **Create Component** tool to build components in-place beginning with **Create In-Place Component** dialog box. See **Figure 18-1**. Enter the name of the new component using the **New File Name** text box. Use the **Template** drop-down list or pick the **Browse Templates** button to select a template file to use for creating the component file. Use the **New File Location** text box or pick the **Browse** button to specify the location of the new, separate component file.

The **Default BOM Structure** drop-down list allows you to adjust the status of the component in the assembly bill of materials. **Figure 18-2** explains each option. Choose the appropriate option depending on how you want the bill of materials to recognize the component. Select the **Virtual Component** check box to create a *virtual component*. Virtual components are not models and do not generate separate component files.

The **Constrain sketch plane to selected face or plane** check box is available for creating a new component in an assembly file that already contains features. When it is selected, you have the option of establishing a constraint between the sketch plane and a selected feature face or plane. Clear the check box if you do not want to specify a constraint between an existing feature plane and the new component. Pick the **OK** button to save the component file and begin component development.

Ribbon
**Assemble
> Component**

Create

Shortcut
Create Component

Type
[N]

virtual component:
An assembly component used primarily to define a separate bill of materials item without creating a model.

Figure 18-1.

Using the **Create In-Place Component** dialog box to specify the characteristics of a new part model built in place. This figure also shows the completed part built in place. Notice the characteristics of the assembly file browser and other components when a part is active.
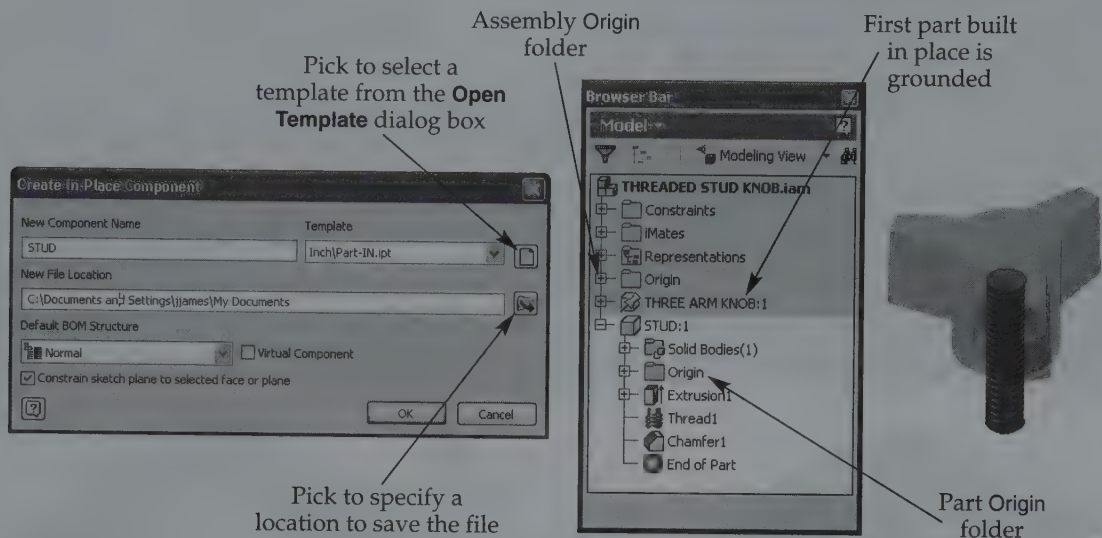
Figure 18-2.

Pick the appropriate bill of materials structure option based on how you want the bill of materials to treat the component.

Option	Result
Normal	Identifies and displays components as assembly components, listed in the order specified in the assembly browser and calculated according to the specified quantity; appropriate for most standard components.
Inseparable	Differentiates welded or otherwise permanently fastened subassemblies from other components. Subassemblies are grouped as parts.
Purchased	Identifies the component as purchased from a vendor; common for standard parts and subassemblies such as switches, fasteners, and springs. Subassemblies group as parts.
Phantom	Component is not displayed in the bill of materials, but exists in the model. Child components do appear in the bill of materials. Phantom components reduce the number of components in the bill of materials or eliminate certain components from the bill of materials.
Reference	Component and child components do not appear in the bill of materials. Assign this option to components that are used for reference purposes and have no other design or bill of materials value.

PROFESSIONAL TIP

You can also adjust the bill of materials structure status while working with individual component files using the **Default BOM Structure** drop-down list in the **Bill of Materials** tab of the **Document Settings** dialog box. Bill of materials settings that are applied to an individual component file automatically update in the assembly bill of materials.

Creating Parts in Place

Figure 18-1 shows the process of creating a part in place by specifying a part file template in the **Create In-Place Component** dialog box. To position the first, or base, part, pick a location in space, or select a plane or the default center point from the assembly Origin folder. If you specify a **Sketch on new part creation** plane in the **Part** tab of the **Application Options** dialog box, a sketch opens on the origin plane associated with the *part*. However, be aware that if you selected an assembly plane to position the part, the assembly and part planes may not correspond. For example, if the **Sketch on new part creation plane** is set to open a sketch on the XZ plane, and you pick the assembly XY plane, the part XZ plane becomes coplanar to the assembly XY plane. This is a common situation. Rarely do assembly and component planes correspond.

If you are using the **No New Sketch** option for new part creation, you can open a sketch on a plane in the part *or* assembly Origin folder. Pick a part plane to associate the sketch with the part. This situation also occurs when you insert an existing part into an assembly. In this case, none of the part sketches link to the assembly or share planes. If you pick an assembly plane to place a sketch, the sketch plane links to the assembly. An adaptive work plane appears at the sketch plane.

NOTE

A base component created in place is grounded just like an inserted base component. Refer again to **Figure 18-1**.

The process of creating additional parts in place is the same as building a base part. However, when existing components are available, you have the option of positioning new parts and opening sketches on a component face or plane. If you open a sketch on a component face or plane, the sketch plane is positioned on an adaptive work plane linked to the selected component face or plane using a flush solution, assembly-mate constraint.

PROFESSIONAL TIP

If you open a sketch plane on a component plane or an assembly origin plane, a flush constraint automatically occurs between the items. If you do not want to establish a constraint, deselect the **Constrain sketch plane to selected face or plane** check box, or pick a location in the graphics window unrelated to features.
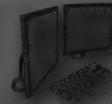
From this point on, creating a part in an assembly file is almost identical to building a part in a part file. However, you can use geometry on other components to help develop new components. For example, use the **Project Geometry** sketch tool to project other component geometry onto a different component sketch plane. You can also end features at other components using **To** and **To Next** extents options. In some ways, it may help to think of other components as other features when modeling in place, but be careful to establish appropriate links between features and components. You will explore other techniques for creating components in place as you learn adaptive modeling later in this chapter.

NOTE

Select the **Mate Plane** check box from the **In-Place Features** area in **Assembly** tab of the **Options** dialog box to form a flush solution, assembly-mate constraint between the component face or plane you select when ending a new feature using the **To** or **From To** extents option.

Ribbon

Model
> Return

Return

Shortcut

Finish Edit

Type

[Ctrl] + [Enter]

When you create or edit a component in place, you adjust the assembly file by adding or editing references to component files. For example, once you create the part, you must finish the assembly edit and re-enter the assembly work environment using the **Return** tool. If you are working on a component in a subassembly, use the **Return to Parent** tool to return to editing the subassembly, or pick the **Return to Top** tool to re-enter the *parent assembly* work environment. You can also finish editing components by double-clicking the parent assembly or a subassembly in the browser.

Exercise 18-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 18-1.

parent assembly: The assembly into which you add parts and subassemblies.

Creating Subassemblies in Place

Figure 18-3 shows the process of creating a subassembly in place by specifying an assembly file template in the **Create In-Place Component** dialog box. A subassembly is an assembly file embedded in the parent assembly file. The process for creating a subassembly is the same as for creating an assembly. Insert parts or subassemblies using the **Place Component** tool, and develop parts and subassemblies in place using the **Create Component** tool. Finish the subassembly edit using the **Return, Return to Parent**, or **Return to Top** tool, depending on the current edit level.

Figure 18-3.

Using the **Create In-Place Component** dialog box to specify the characteristics of a new subassembly model built in place. This figure also shows the completed part subassembly in place. Notice the characteristics of the assembly file browser and other components when a subassembly is active.
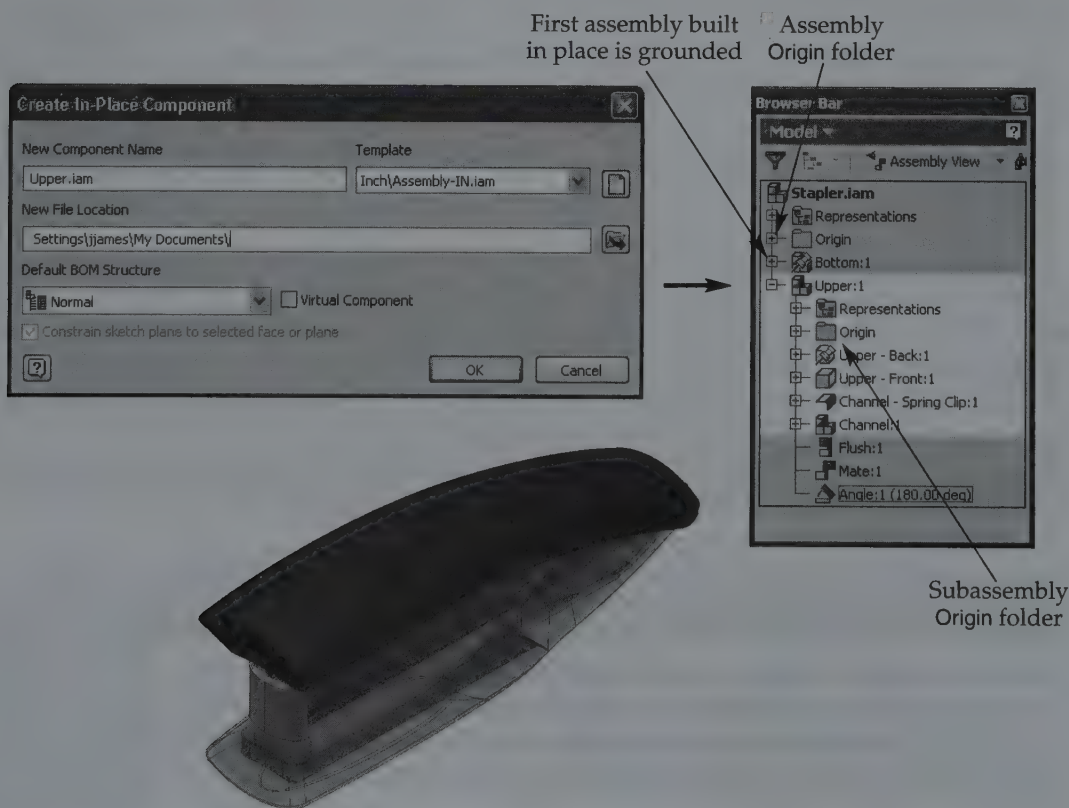
When building a subassembly in place, you must be constantly aware that you are creating parts and assemblies or subassemblies in an assembly or subassembly that is in the parent assembly. Ensure that the subassembly remains active while you are inserting and creating subassembly components.

Exercise 18-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 18-2.

Demoting and Promoting Components

An alternative for creating subassemblies is to insert or create separate components inside the parent assembly file without creating a subassembly, and then *demote* the components. Select the components to group from the browser or the graphics window, right-click, and select **Demote**, or select the components and press [Tab]. The **Create In-Place Component** dialog box opens, allowing you to specify the subassembly file name and location. For the opposite application, you can *promote* components. Right-click on the subassembly in the browser or in the graphics window and select **Promote** from the **Component** cascading submenu. Another option is to select the subassembly and press [Shift] and [Tab]. The subassembly components become individual components in the main assembly.

You can also place components from the parent assembly file into a subassembly by dragging the component from its current location in the browser to a location under the **Origin** folder of the subassembly. To place a subassembly component into the parent assembly file, drag the component from the subassembly and insert it under the **Origin** folder of the parent assembly. If you place a component under the **Origin** folder of a subassembly, it is inserted in the subassembly.

demote: Group more than one part in an assembly to create a subassembly.

promote: Remove parts from a subassembly and make them individual parts in the parent assembly.

Dragging and dropping components into and out of assemblies and subassemblies can become confusing. To confirm component locations, collapse all child nodes, then open and close individual subassemblies as needed. Use the **Find Component** tool to locate components in the browser and in the graphics window.

Ribbon

Tools
> Find

Find

Adaptive Modeling

Inventor includes adaptive capabilities that help automate revisions and preserve design intent. In order to change the shape of a *rigid body*, you must manually edit feature parameters, which does not ensure parametric relationships between assembly components. In order for a component to adapt to design changes, you must specify the component *and* the actual items that adapt, such as sketch geometry and feature parameters, as adaptive. **Figure 18-4** shows an example of how adaptive assembly components adapt to design revisions.

rigid body: A feature that is not adaptive.

Figure 18-4.
A hinge assembly with adaptive components. This example shows how adaptive components adjust automatically to significant changes made to material and size parameters.

Original Design | Design Revision
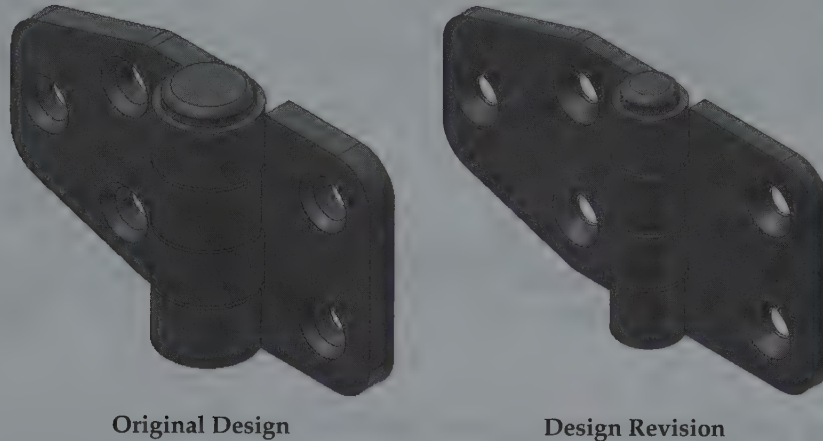
In-Place Adaptivity

By default, some adaptivity occurs while you are building components in place, as you reference other components or the items associated with the parent assembly. For example, when you open a sketch on the face of a different component, an adaptive work plane automatically forms linking the sketch plane with the component face. The location of the sketch plane adapts with changes to the component face. See **Figure 18-5**.

The **In-Place Features** area in **Assembly** tab of the **Options** dialog box includes settings that allow you to build adaptive components in place, if appropriate. Select the **Mate Plane** check box to form an assembly-mate constraint between the component face or plane you select when you end a new feature using the **To** or **From To** extents option. Pick the **Adapt Feature** check box to adapt features created using the **To** or **From To** extents option when changes occur to the selected component face or plane. See **Figure 18-6**. If you do not select the **Adapt Feature** check box, a work plane forms at the original extents of the feature.

Choose the **Enable Associative Edge/Loop Geometry Projection During In-Place Modeling** check box to project *and* link geometry from a component onto the new component sketch plane when using tools such as **Project Geometry**. When you create associated sketch geometry, the new component, sketch, and feature automatically

Figure 18-5.
An adaptive work plane locates a new feature sketch plane on another component face. The adaptive work plane, sketch, and feature adjust to changes made to the component face. Notice the icons indicating the adaptive items.

Adaptive work plane

Sketch plane adapts

.500

Addaptive icons

Selected existing component face
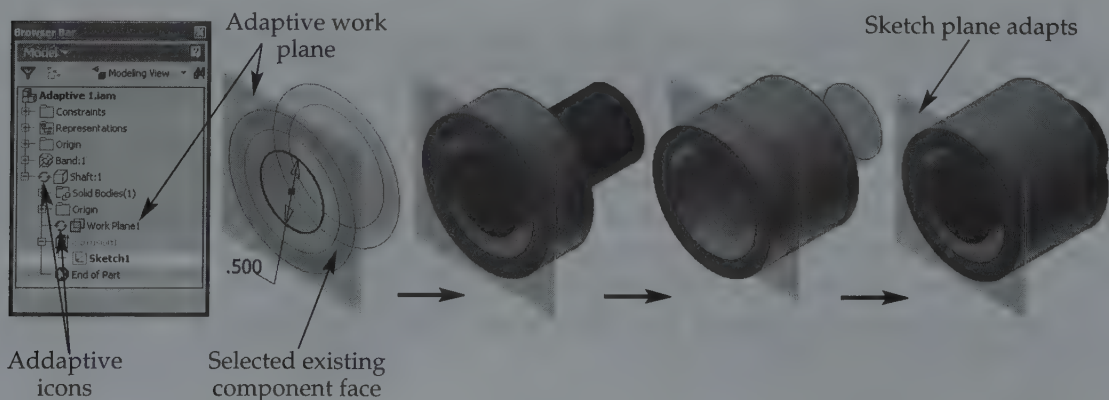
Figure 18-6.
Select the **Adapt Features** check box to adapt features that extend to or from and to other component faces or planes.

Adaptive feature

Face selected to end extrusion

Feature does not adapts to design changes

Feature adapte to design changes

Original Extrusion

☐ Adapt feature

☑ Adapt feature
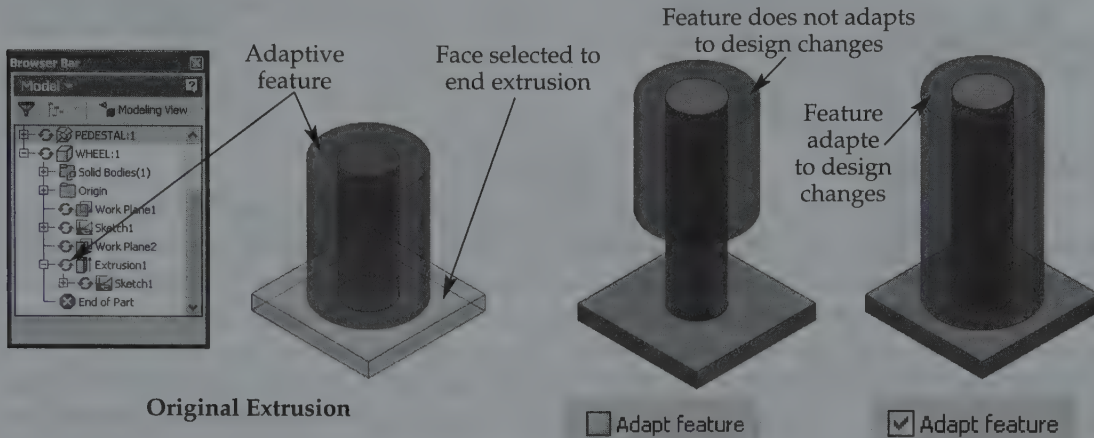
become adaptive, and reference geometry appears in the browser. If you deselect the check box, you can still project component geometry onto the sketch plane, but the projection does not adapt to design changes. See **Figure 18-7**.

> **NOTE**
>
> Right-click on a reference in the browser and select **Break Link** to remove the relationship. Sketch geometry remains, but the sketch no longer looks to the reference component or feature to acquire data, allowing you to add constraints if necessary.

Figure 18-7.
The **Enable Associative Edge/Loop Geometry Projection During In-Place Modeling** check box is active by default and allows you to project *and* link geometry from a component onto the new component sketch plane when using tools such as **Project Geometry**.

References listed in the browser (6 projected edges)

Projected geometry linked to component

Component updated according to associated geometry edit

☑ Enable associative edge/loop geometry projection during in-place modeling

Projected geometry used for sketch development but not linked to component

Component is free to move

Component is not associated with projected geometry

☐ Enable associative edge/loop geometry projection during in-place modeling
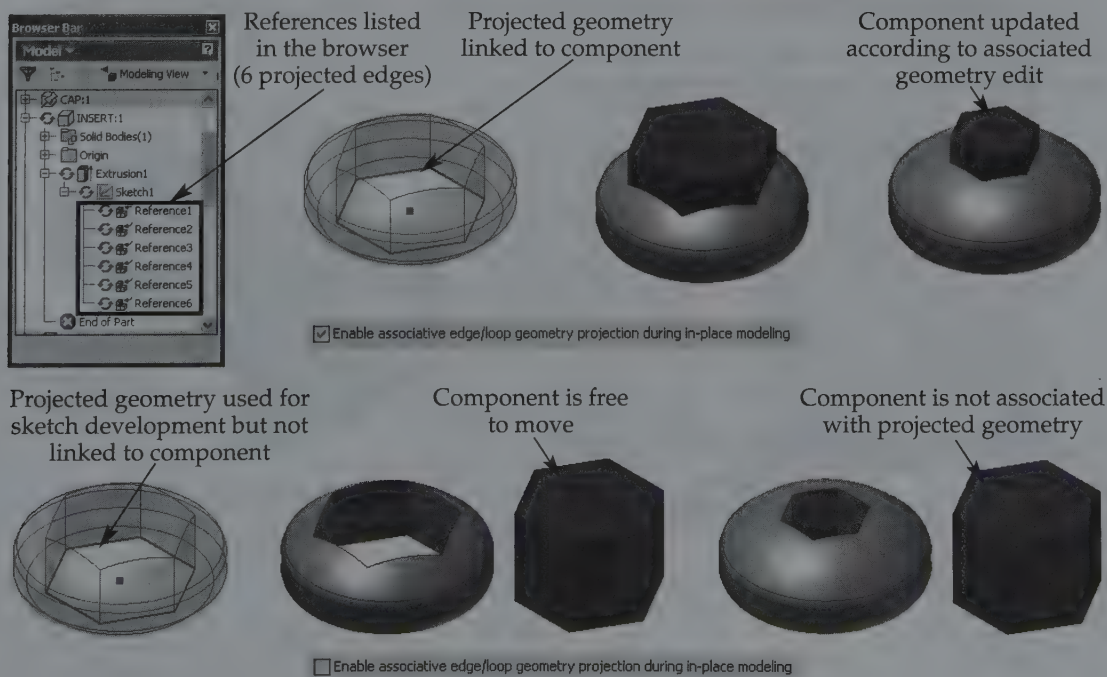
Exercise 18-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 18-3.

Adaptive Sketches

Use an *adaptive sketch* to adjust sketch parameters according to other components. You typically fully constrain a sketch to ensure correct dimensions and geometry. However, the key to developing a feature with an adaptive sketch is to under-constrain the sketch. **Figure 18-8** shows a sketch created without any dimensional constraints, made adaptive, and then extruded. When added to an assembly, the sketch is free to constrain to other component geometry using assembly constraints, in this example mate solution, mate constraints. You can also omit geometric constraints as needed to provide adaptability.

Most adaptive sketches still include some or most constraints. You must determine which sketch items to define and which to adapt, depending on the design. To make a non-adaptive sketch adaptive, right-click on the sketch in the browser or graphics window and select **Adaptive**. Any features consuming the sketch automatically become adaptive. A sketch also becomes adaptive if you make a sketched feature adaptive.

Adaptive Features

You can adapt features using adaptive sketch geometry and/or by adapting non-sketch parameters, such as an extrusion distance or revolution angle, using assembly constraints. In order for a feature to adapt, you must specify the feature as adaptive. **Figure 18-9** shows an assembly that requires a Ø.5″ rod of an unknown length that may change during the design. By specifying the extrusion in the rod part as adaptive, you can constrain one end of the rod to the other component, and then use an appropriate constraint, in this example an assembly-insert or flush solution assembly-mate constraint, to develop the correct rod.

Figure 18-8.
An example of using an under-constrained, adaptive sketch to create a part that adapts to another component using assembly-mate constraints.

Adaptive Sketch
(no dimensional constraints)

Component

Constrained extrusion

Original extrusion
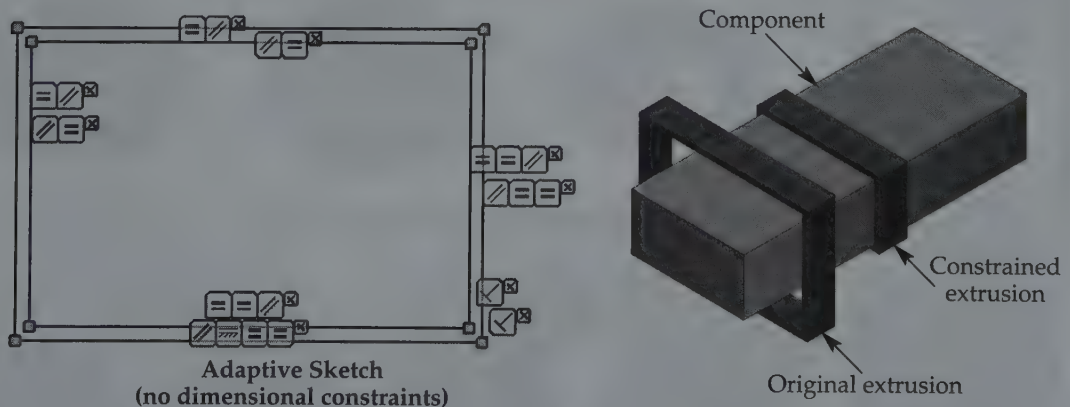
Figure 18-9.
An example of using an adaptive extrusion to create a part that adapts to another component using assembly-mate constraints.

Adapted extrusion

.500

Sketch   Original extrusion   Insert constraint   Insert constraint
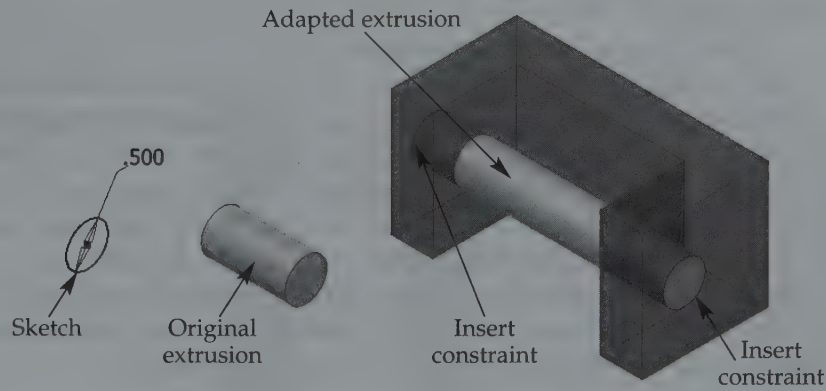
Features that you can set as adaptive include extrusions, revolutions, holes, and work features. To make a feature adaptive, if it does not happen automatically, right-click on the feature in the browser or graphics window and select **Adaptive**. This method makes all elements of the feature adaptive. The part consuming the feature and any sketches consumed by the feature become adaptive. To make certain extrusion, revolution, or hole elements adaptive, right-click on the feature in the browser or graphics window and select **Properties** to access the **Feature Properties** dialog box. See **Figure 18-10**. Select the check boxes corresponding to only those items you want to make adaptive.

> **NOTE**
>
> If you check **Features are initially adaptive** in the **Assembly** tab of the **Options** dialog box, all features that can be adaptive become adaptive when they are created.

Figure 18-10.
A—The **Feature Properties** dialog box for an extrusion. B—The **Feature Properties** dialog box for a revolution. C—The **Feature Properties** dialog box for a hole.

A   B   C
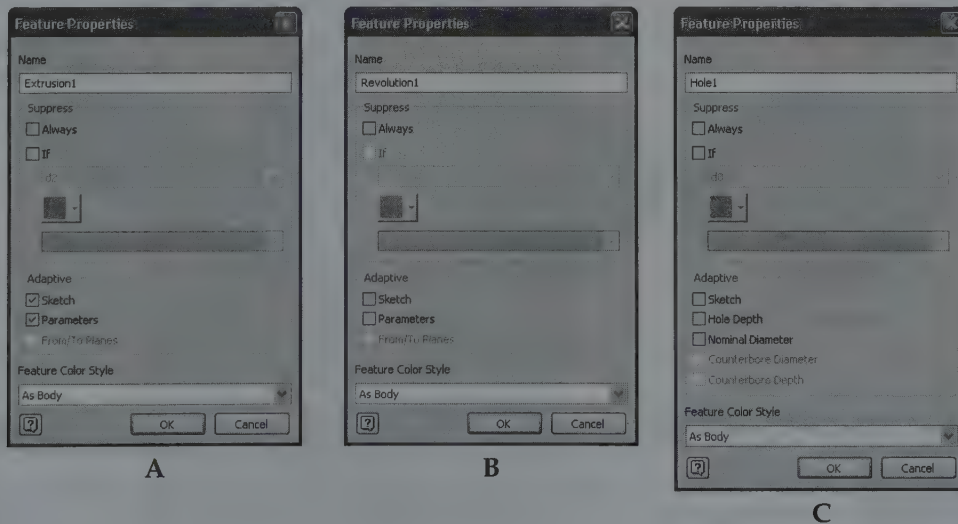
Adaptive Parts

In order for a part to adapt, you must specify the part as adaptive, and it must include adaptive features. In addition, unless you pick the **Adaptively Used in Assembly** check box in the **Modeling** tab of the part's **Document Settings** dialog box, even if you identify a part as adaptive in the part environment, you must specify the part as adaptive after you insert the part into an assembly. To make a non-adaptive part adaptive, right-click on the part in the browser or graphics window and select **Adaptive**.

You can also right-click on the part in an assembly and select the **iProperties** shortcut menu option to access the part **iProperties** dialog box. See **Figure 18-11**. Pick the **Occurrence** tab, and then select the **Adaptive** check box to make the part adaptive. **Figure 14-12** shows an example of using an adaptive part.

Exercise 18-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 18-4.

Adaptive Subassemblies

In order for a subassembly to adapt, you must specify the subassembly as adaptive, and it must include adaptive components that include adaptive features. Adapt a subassembly using techniques similar to those for adapting parts, such as editing associated component geometry or using assembly constraints. To make a non-adaptive subassembly adaptive, right-click on the subassembly in the browser or graphics window and select **Adaptive**.

You can also right-click on the subassembly in the assembly and select the **iProperties** shortcut menu option to access the part **iProperties** dialog box. Pick the **Occurrence** tab, and then select the **Adaptive** check box to make the subassembly adaptive. **Figure 14-13** shows an example of using an adaptive subassembly.

Figure 18-11.
The **iProperties** dialog box for a part, **Occurrence** tab.
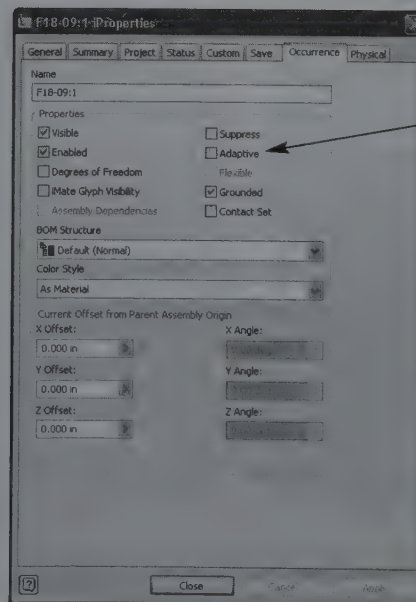
Adaptive check box

Figure 18-12.
An example of creating and using an adaptive part.

.500

No dimension

Initial
Adaptive
Sketch

Adaptive
Hole
Sketch

Final
Part

Edit extrusion
distance

Part adapts to
new extrusion

Initial Design

New Design
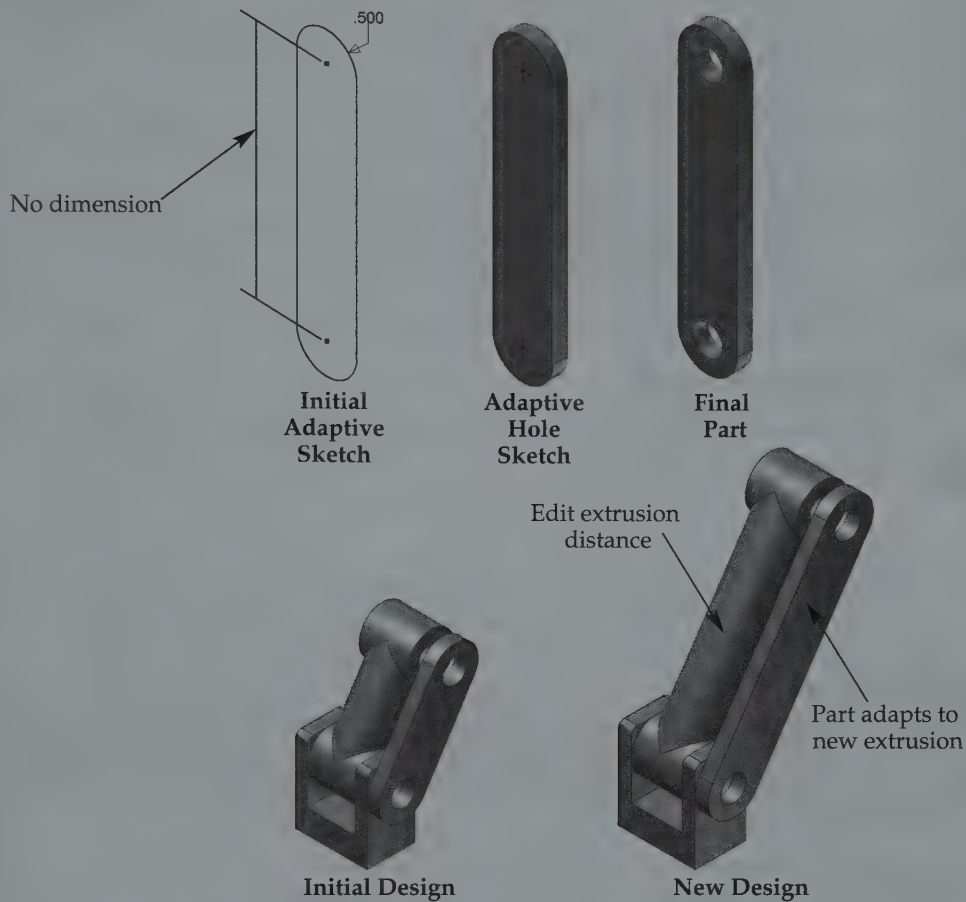
Figure 18-13.
An example of creating and using an adaptive subassembly.

Subassembly

Subassembly adapts to
new bracket design

Subassembly

Assembly
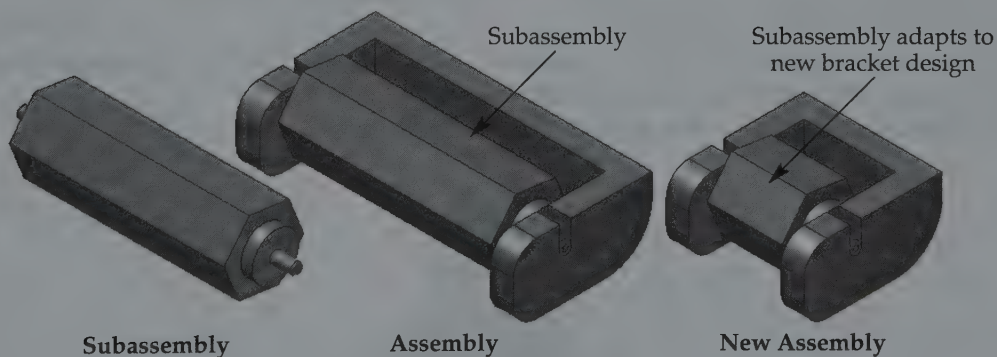
New Assembly

NOTE

If an assembly contains several of the same adaptive components, only one of the occurrences adapts to other component geometry. As a result, the original components change to reflect the adapted design. To overcome this problem, save a copy of the parent and child components, using different names. You may also want to save separate unadaptive copies.
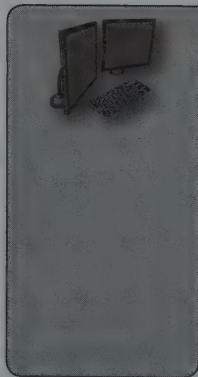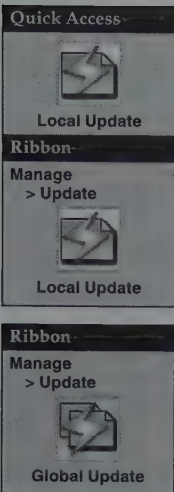
PROFESSIONAL TIP

If you have difficulty constraining adaptive components and receive the "constraint is inconsistent with another" error message, consider the following actions:

- Remove conflicting sketch constraints.
- Ensure that the appropriate sketches, features, parts, and subassemblies are adaptive.
- Try using an alternative constraint.
- Try using the **Local Update** and **Global Update** tools, repeatedly if necessary.

Exercise 18-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 18-5.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. What is an adaptive component?
2. Name the tool used to build components in place.
3. What is a virtual component?
4. Describe a parent assembly.
5. Define *demote*.
6. Define *promote*.
7. Briefly describe a rigid body.
8. Describe what you must do in order for a component to adapt to design changes.
9. Explain the function of an adaptive sketch.
10. How do you make a non-adaptive sketch adaptive?

Problems

- Insert existing components, prepare new components for insert, or build components in place as instructed to create the following assemblies.
- Use appropriate sketching techniques and fully constrain sketch geometry.
- Add all necessary assembly constraints.
- Use your own judgment and approximate dimensions when necessary.
- Add as much information as possible to the **iProperties** dialog box for component and assembly files. Assign the specified material and color to parts.
- Follow the specific instructions for each problem to create the assemblies.

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

1. **Title:** PUSH PIN (IN-PLACE)
   Units: Inch
   Template: Assembly-IN.iam
   Part Number: IAA-104
   Project: PUSH PIN (IN-PLACE)
   Save as: P18-1.iam
   Specific Instructions: Insert the push pin head, P6-7.ipt. Create a pin in place using the Part-IN.ipt template and a file name P18-1.ipt. To build the pin:
   A. Locate the part on the face shown, and then open a sketch on the same face. Project the edge of the hole in the head onto the sketch plane. Extrude the sketch to the depth of the hole in the head using the **To** extents option.
   B. Open a sketch on the face of the extrusion exposed at the head, and project the extrusion edge. Do not project the edge of the hole in the head. Extrude the sketch .375″ in the positive direction and then chamfer the extrusion edge to create a point using a .15″ × .02″ chamfer.
   C. In the **iProperties** dialog box, specify a title of PIN (IN-PLACE) and a part number of IAA-104-02, name the project PUSH PIN (IN-PLACE), and specify the Steel, High Strength Low Alloy material.
   D. Change the color to Chrome and finish the part edit.

   Resave the assembly and all dependents. The final assembly should look like the assembly shown.

Selected face

Basic

2. **Title:** FORK
   Units: Metric
   Template: Assembly-mm.iam
   Part Number: IAA-025
   Project: FORK
   Save as: P18-2.iam
   Specific Instructions: Insert the fork body, P12-5.ipt. Create a fork handle in-place using the Part-mm.ipt template and a file name P18-2.ipt. To build the handle:
   A. Locate the part on the face of the fork body shown, and then open a sketch on the same face. Project the furthest edges of the fork body onto the sketch plane. Offset the projected fork loop 3 mm away from the fork, and extrude the handle profile 90 mm along the fork.
   B. In the **Properties** dialog box, specify a title of FORK HANDLE and a part number of IAA-0025-02, name the project FORK, and specify the default material.
   C. Change the color to Wood (Ash) and finish the part edit.
   Resave the assembly and all dependents. The final assembly should look like the assembly shown.

3 ← Offset distance

Projected from edge of P12-5.ipt (fork)

Selected face
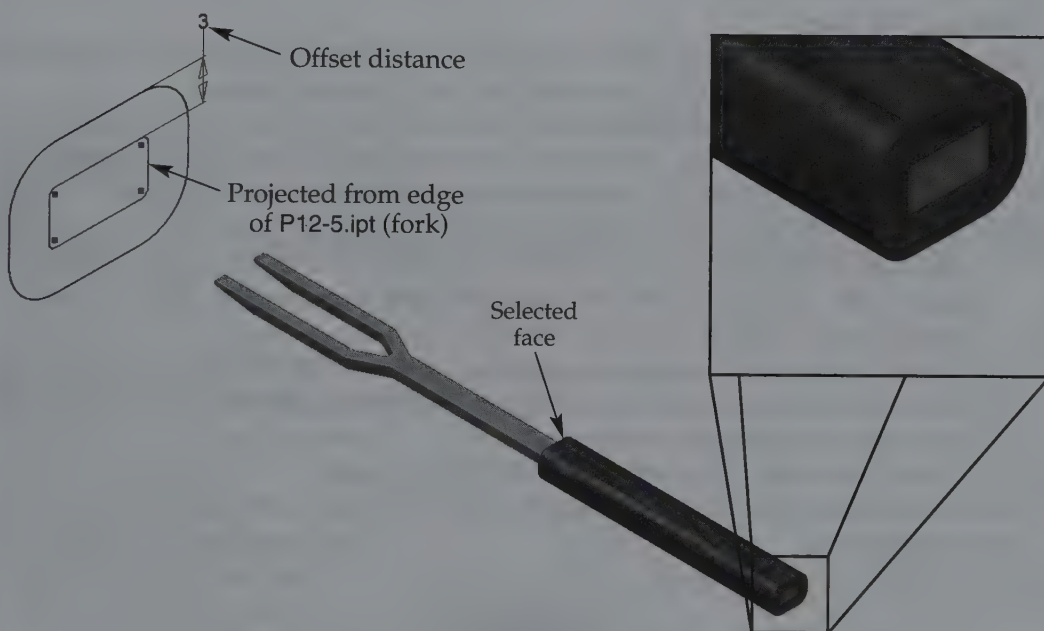
3. **Title:** HOUSING ASSEMBLY
   Units: Inch
   Template: Assembly-IN.iam
   Part Number: IAA-039
   Project: HOUSING
   Save as: P18-3.iam
   Specific Instructions: Insert the housing, P16-2.ipt. Create a housing cover in place using the Part-IN.ipt template and the file name P18-3.ipt. Use the 11 GAGE STEEL sheet metal rule, a material set according to the sheet metal rule, and the Galvanized (texture) color. Use correct projection and adaptivity techniques to ensure that as changes occur to the housing part, the cover will adapt without errors. Resave the assembly and all dependents. The final assembly should look like the assembly shown.
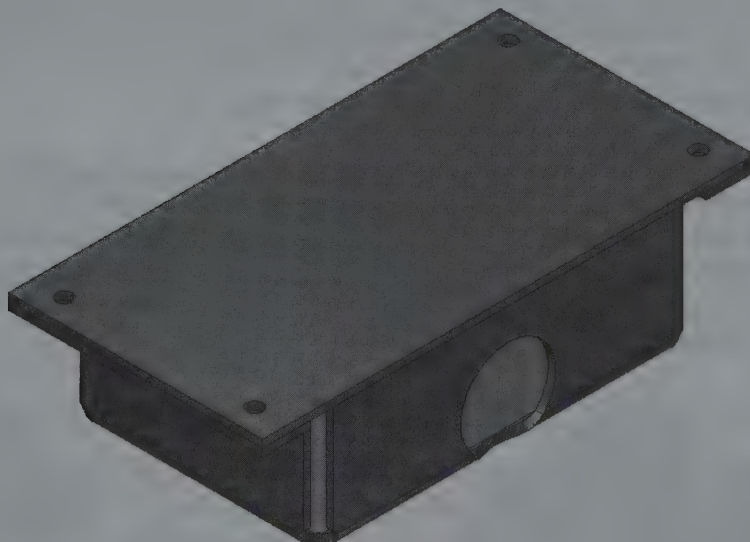
4. **Title:** ROTATING HINGE ASSEMBLY
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-042
   Project: ROTATING HINGE ASSEMBLY
   Save as: P18-4.iam

   Specific Instructions: Design a rotating hinge assembly similar to the example shown. Assign iMates to each component and then use the iMates for assembly. To build each component:

   A. Create the housing shown in place using dimensions of your choice. Save the part as P18-4a.ipt. In the **iProperties** dialog box, specify a title of HOUSING and a part number of IAA-042-01, name the project ROTATING HINGE ASSEMBLY, and specify the Stainless Steel material.

   B. Create the bushing shown in-place. Design the bushing to fit inside the housing as shown. Save the part as P18-4b.ipt. In the **iProperties** dialog box, specify a title of BUSHING and a part number of IAA-042-02, name the project ROTATING HINGE ASSEMBLY, and specify the Rubber material.

   C. Insert another instance of P18-4b.ipt.

   D. Create the pin shown in place. Design the bushing to fit inside the housing as shown. Save the part as P18-4c.ipt. In the **Properties** dialog box, specify a title of PIN and a part number of IAA-042-03, name the project ROTATING HINGE ASSEMBLY, and specify a Stainless Steel material. Change the color to Chrome.

   Resave the assembly and all dependents. The final assembly should look similar to the assembly shown.

P12-3A.ipt

P12-3B.ipt

P12-3C.ipt
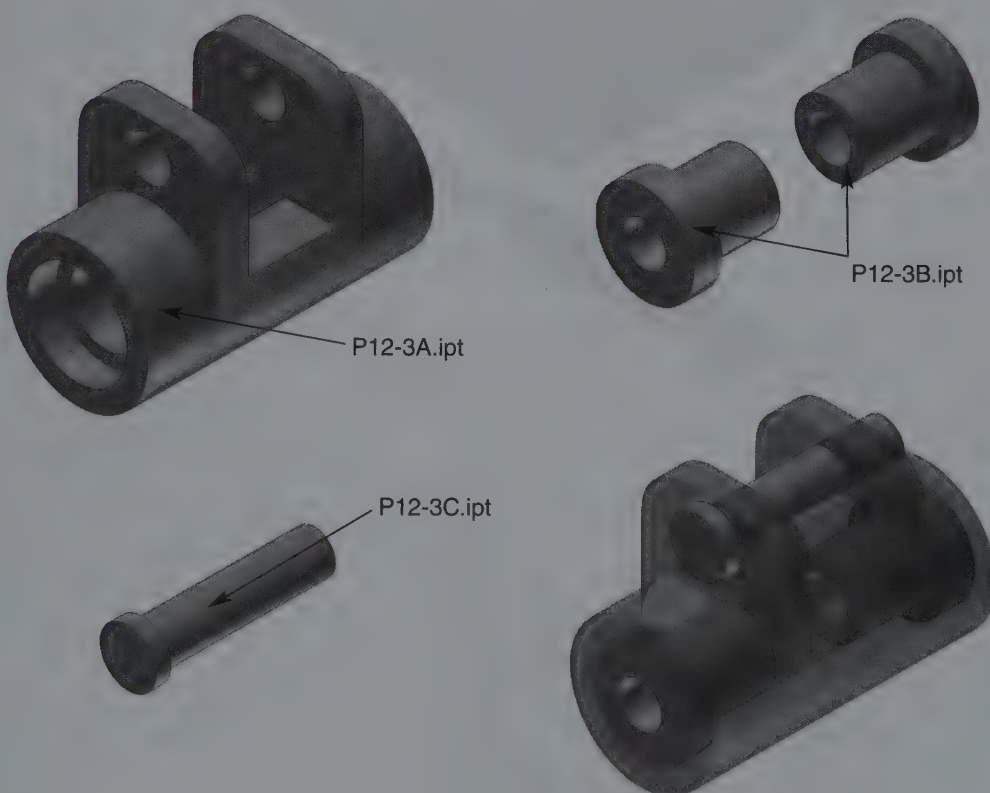
Additional Assembly Tools

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Pattern, mirror, and copy components.
✓ Edit assemblies and adjust component colors.
✓ Replace, move, and rotate components.
✓ Apply assembly section views.
✓ Create flexible subassemblies and assembly representations.

Inventor includes many assembly tools and options in addition to those described in Chapters 17 and 18. This chapter describes assembly tools that allow you to pattern, mirror, and copy components. You will also learn about options for adjusting and managing assemblies.

Patterning Components

Figure 19-1 shows an example of using a *component pattern* to arrange multiple copies of the same components, in this case fasteners. This technique is less time-consuming than inserting and individually constraining each component. Access the **Pattern Component** tool to pattern components using the **Pattern Component** dialog box. See **Figure 19-2**.

component pattern: A reoccurrence of components in a designated configuration.

Ribbon

**Assemble
> Component**

Pattern

Shortcut

**Component
> Pattern**

Associative Patterning

Use the **Associative** tab, shown in **Figure 19-2**, to pattern components in relation to an existing component feature pattern. For example, the pattern of fasteners shown in **Figure 19-1** are associated with a circular feature pattern of holes in the cover plate and elbow. The **Component** button is active by default, allowing you to choose components to pattern. The component in **Figure 19-2** is a constrained screw.

Pick the **Associative Feature Pattern Select** button and select the feature pattern to reference. The associative feature pattern in **Figure 19-2** is a rectangular pattern of holes on the slide bar hinge component. The name of the pattern appears in the **Feature Pattern Select** display box, and Inventor previews of the pattern operation. If the preview looks correct, pick the **OK** button to generate the pattern.
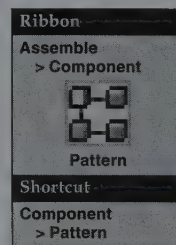
Figure 19-1.
An example of an assembly with a pattern of nut and bolt fastener components.
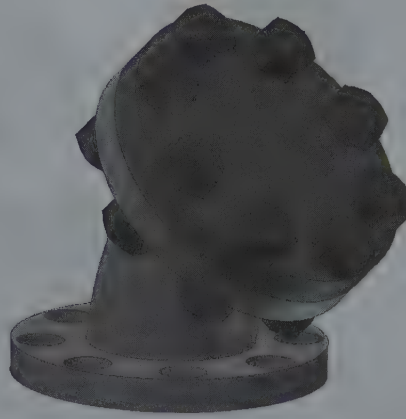
Figure 19-2.
The **Associative** tab of the **Pattern Component** dialog box. In this example, the slide bar hinge component includes a feature pattern and the assembly contains an associative component pattern.

Pick to select a component to pattern

Rectangular tab

Circular tab

Screw

Feature pattern

Associative component pattern

Feature Pattern Select

Rectangular Pattern1

Associative tab

Pick to select a feature pattern to reference
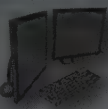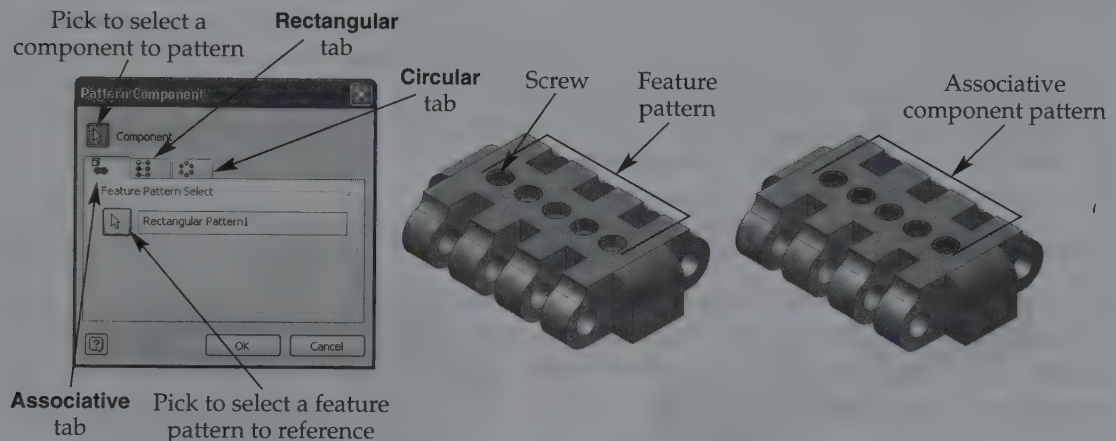
PROFESSIONAL TIP

The associative function of the **Pattern Component** tool is excellent for quickly adding components. The assembly pattern adjusts parametrically to any changes made to the feature pattern.

Exercise 19-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-1.

Rectangular Patterning

Select the **Rectangular** tab, shown in **Figure 19-3**, to create a rectangular pattern of a component without identifying an associative feature pattern. For example, **Figure 19-3** shows patterning a grounded bracket that is later linked with other components created in-place. The **Component** button is active by default, allowing you to choose components to pattern.

Figure 19-3.
The **Rectangular**
tab of the **Pattern
Component** dialog
box and an example
of a rectangular
component pattern.
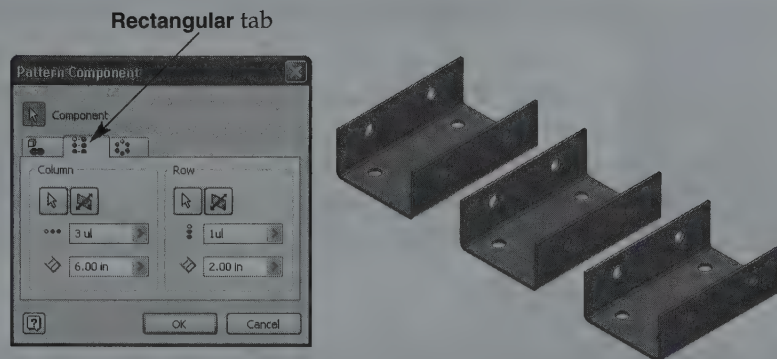
Pick the **Column Direction** button and select a feature edge, axis, or work axis, including an axis in the **Origin** folder of the browser, to specify the column direction. If the preview and direction arrow appear incorrect, pick the **Flip** button to reverse the direction. Specify the number of occurrences using the **Count** text box. Then specify the spacing using the **Spacing** text box. For example, to pattern a block that is 1″ wide and include a 1/2″ space between pattern copies, you must specify a 1 1/2″ spacing.

After you define **Column** options, repeat the steps to define **Row** settings. For most applications, the row direction is nonparallel to the column direction. Direction arrows and a preview display the complete pattern operation. Pick the **OK** button to generate the rectangular pattern.

Circular Patterning

Select the **Circular** tab, shown in **Figure 19-4**, to create a circular pattern of components without identifying an associative feature pattern. For example, **Figure 19-4** shows circular patterning a constrained stud, which will later be welded to the plate. The **Component** button is active by default, allowing you to choose components to pattern.

Pick the **Axis Direction** button and select a feature edge, axis, work axis, or an axis in the **Origin** folder of the browser to specify the axis around which components pattern. If the preview and direction arrow appear incorrect, pick the **Flip** button to reverse the direction. Specify the number of occurrences using the **Count** text box. Then use the **Angle** text box to set an included angle to divide the count equally between. For example, to distribute the copies around a 360° circle, divide 360 by the count to identify the angle. Pick the **OK** button to generate the circular pattern.

Figure 19-4.
The **Circular** tab
of the **Pattern
Component** dialog
box and an example
of a circular
component pattern.
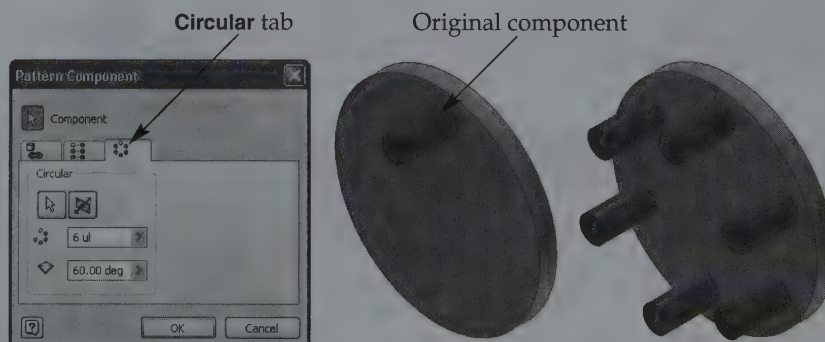
Mirroring Components

mirrored components: Reversed copies of a component.

Ribbon

Assemble > Component

Mirror

Shortcut

Component > Mirror

mirror plane: A plane of symmetry about which components are mirrored.

derived body: A single body, or object, extracted from multiple features or components.

Mirrored components can save significant time by eliminating the need to create additional components using traditional techniques. **Figure 19-5** shows an example of mirroring components to create an assembly. In this example, the first mirror operation creates the symmetrical legs and provides reference geometry for creating the upper bracket.

Access the **Mirror Component** tool to mirror components using the **Mirror Components** dialog box. See **Figure 19-6**. The **Components** button is active by default, allowing you to select components to mirror. Then pick the **Mirror Plane** button and select the *mirror plane*. Choose any available plane, including a planar feature face, work plane, or a plane in the **Origin** folder of the browser.

Selected components are listed under the assembly file in the list box. The default **Mirror** symbol indicates that the mirrored component becomes a *derived body* in a separate file, which is appropriate when the mirror results in a unique component. To reuse the exact component, without creating a separate file, select the **Copy** symbol to change it to a **Reuse** symbol. For example, you do not need to create a new mirrored copy of a standard screw. If you do not want to mirror *or* reuse a component, pick the **Mirror** symbol until you see the **Excluded** symbol.

Pick the **More** button to access additional mirroring preferences and preview options. Select the **Reuse Content Library Components and Factory Parts** check box to reuse, not mirror, library components such as shared fasteners or springs. The **Preview Components** area contains check boxes that allow you to define the type of components that will be previewed in the graphics window during the mirror process.

Figure 19-5.
An example of mirroring components to create an assembly.

Created in place by referencing the first mirror

Second mirror

First mirror
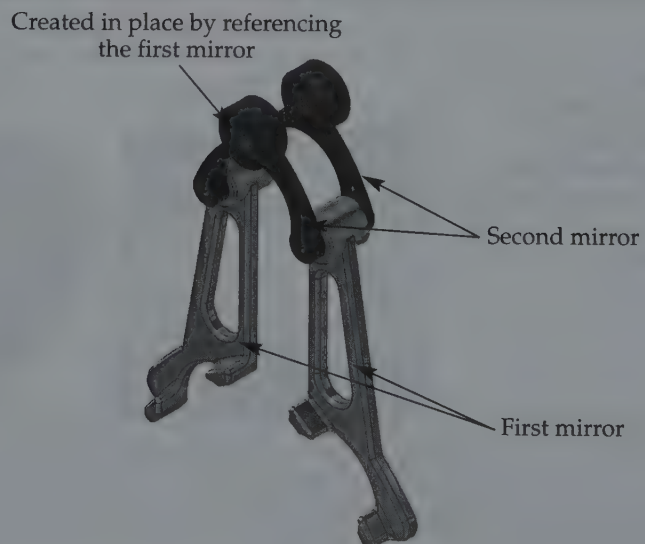
Figure 19-6.
The **Mirror Components** dialog box and an example of selecting components to mirror and a mirror plane.

Mirror button **Reuse** button **Exclude** button

Selected mirror plane (offset work plane)

Selected assembly component

Preview of mirrored component
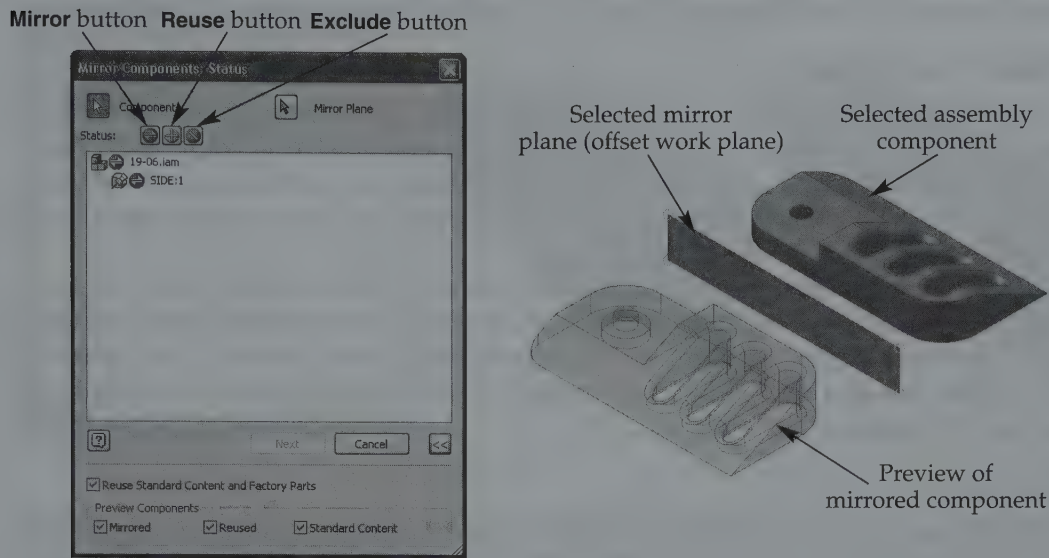
Name Files Dialog Box

Pick the **Next** button to access the **Mirror Components: File Names** dialog box shown in **Figure 19-7**. Remember, mirrored components are copied to new files as derived bodies. Use the **File Names** dialog box to define file name and location preferences for the new files created when you mirror components. If necessary, pick the **Return to Selection** button to reopen the **Mirror Components** dialog box and redefine selections. Pick the **OK** button to generate the mirrored components and the separate component files. The selected **Component Destination** radio button controls whether you reenter the assembly work environment, or if the mirror occurs in a separate assembly file.

Naming Files

The **Name** column lists each selected component nested in the parent assembly file. The **New Name** column lists the default name assigned to the new files created during the mirror operation. The **Naming Scheme** area specifies default new file name characteristics. Select the **Suffix** check box to add the characters in the **Suffix** text box after the existing file name. Select the **Prefix** check box to add the characters in the **Prefix** text box before the file name.

Figure 19-7.
The **Mirror Components: File Names** dialog box.
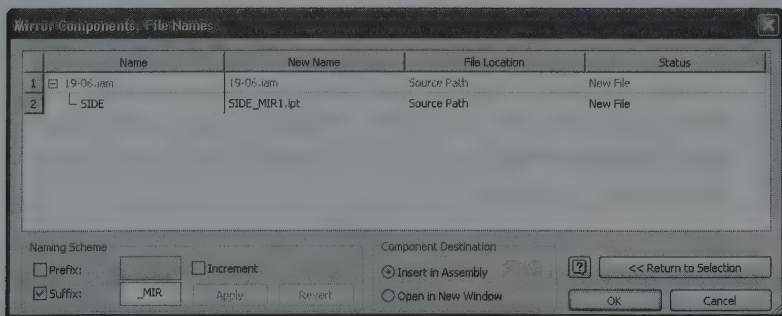
Pick the **Apply** button to observe changes made to the naming scheme in the **New Name** column. To undo the settings and return to the previous naming scheme, pick the **Revert** button. Pick the **Increment** check box to apply incremental numbers to file names. You can also select the default new name and modify the name using the text box.

Locating Components

The **File location** column identifies the location of each new file. The default **Source Path** option places the new file in the same location as the existing component. Right-click on **Source Path** and select **Workspace** to save the file to the current workspace. Right-click on **Source Path** and pick **User Path** to place the file in a different location using the **Browse** button to specify a path.

The **Component Destination** area allows you to define where to place mirrored components. Select the **Insert in Assembly** radio button to return to the graphics window and place the mirrored components into the current assembly file. Select the **Open in New Window** radio button to create a new assembly file and add only the mirrored components to the new file.

File Status

The **Status** column identifies the status of the mirror process based on the specified new file names and locations. A **New File** status identifies that the new file name is acceptable and does not already exist. A **Reuse Existing** status indicates that you can use the new part file name, but an existing file already uses the same name. If you enter a name for an assembly file that is the same as an existing assembly file name, an unacceptable **Name Conflict** status occurs and the background is red.

Exercise 19-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-2.

Copying Components

For most applications, insert additional copies of existing components using one of the techniques described in Chapter 17, or pattern features when appropriate. However, a different operation, conducted using the **Copy Components** tool, is required to reuse a component but also make changes to the component without affecting the component as used in other assemblies. Copying components accelerates the traditional process of saving copies of files. **Figure 19-8** shows an example of copying components to create a unique hinge design without overriding the original design.
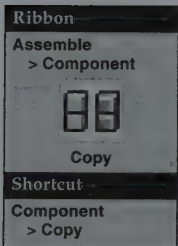
> **NOTE**
>
> Do not confuse the process of inserting, or reusing, existing component files into an assembly with copying components using the **Copy Components** tool. When you copy a component, you create a completely new file.

The **Copy Components** dialog box appears when you access the **Copy Components** tool. See **Figure 19-8**. The **Components** button is active by default, allowing you to pick components to copy. Selected components are listed under the assembly file in the list box. The default **Copy** symbol indicates that a copy of the component forms as

Figure 19-8.
Using the **Copy Components** dialog box to copy hinge components as separate files, and then modifying the copies to create a unique product.
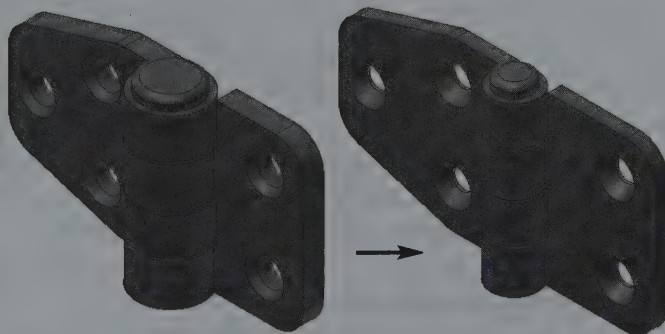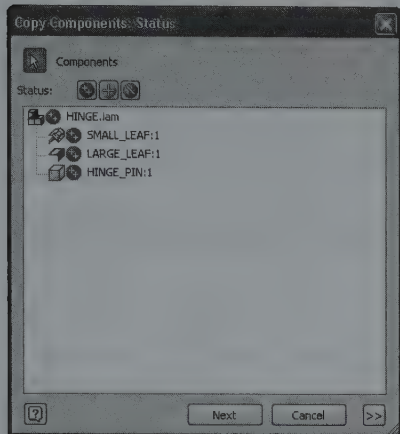
a separate component file, which is appropriate if you plan to edit the copy. To reuse the exact component, without creating a separate file, select the **Copy** symbol to change it to a **Reuse** symbol. For example, you do not need to create a new file of a standard screw. If you do want copy *or* reuse a component, pick the **Copy** symbol until you see the **Excluded** symbol.

Pick the **More** button to access the **Reuse Content Library Components and Factory Parts** check box to reuse, not copy, library components such as shared fasteners or springs. Pick the **Next** button to access the **Copy Components: File Names** dialog box. This is the same dialog box that appears when you mirror components using the **Mirror Components** tool. Use the options in the **Copy Components: File Names** dialog box to define file name and location preferences for the new component files created when you copy components. Depending on the selected **Component Destination** radio button, pick the **OK** button to re-enter the assembly work environment and pick a location to place the copied assembly component, or display the copies in a new assembly file.

NOTE

Mirrored and copied components are grounded in space, but they contain no other assembly constraints.

Exercise 19-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-3.

Edit a component file to make changes to component parameters. You can open a component from within an assembly by right-clicking on the component in the browser or graphics window and selecting **Open**. You can also edit a component in place, which is what you do when you build components in place. Editing a component in place is often the easiest way to make changes and explore design options in relation to the assembly. To edit a component in place, double-click on the component in the browser or graphics window, right-click on the component and select **Edit**, or select the **Modeling View** option in the browser. Several other tools and options are also available for adjusting assembly components.

Replacing Components

The **Replace Component** and **Replace All** tools provide an effective way to replace components without going through the process of deleting, inserting, and re-constraining the new components. Access the **Replace Component** tool to replace a single component. Access the **Replace All** tool to replace all the instances of a component. The **Open** dialog box appears when you select a component to replace, allowing you to locate the replacement. The replacement is positioned and changed according to the original component, including changing to the same color override. However, if the new component geometry is significantly different from the old, as shown in **Figure 19-9**, constraints may no longer exist.

Ribbon
Assemble
> Component

Replace

Shortcut
Component
> Replace

Type
[Ctrl]+[H]

Ribbon
Assemble
> Component

Replace All

Shortcut
Component
> Replace All

Type
[Ctrl]+[Shift]+[H]

Exercise 19-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-4.

Figure 19-9.
An example of replacing a component with one that is similar to the original and with a very different design. Constraints may automatically be removed if the replacement is significantly different.

Replacement Similar
to Original

Replacement Very Different
from Original

Moving Components

When you drag an unconstrained component, the component remains in the moved location unless you move it back, undo the move, or add constraints. Use the **Move Component** tool to move a constrained, or even grounded, component for clarity and display purposes. Hold down the left mouse button on the component, drag the component, and release the left mouse button to drop the component. Continue moving components as required. To exit, press [Esc], right-click and pick **Done**, pick anywhere in space, or access another tool. See **Figure 19-10**.
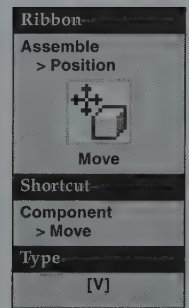
Rotating Components

Often while working with an assembly, you will find that a component is not in the correct orientation to begin constraining. You may also need to orbit a specific component without changing the view of the remaining components. Use the **Rotate Component** tool to orbit an unconstrained, constrained, or even grounded component. Pick the component to rotate and orbit the component as you would the model using the **Free Orbit** tool. Continue rotating components as required. To exit, press [Esc], right-click and pick **Done**, pick anywhere in space, or access another tool. See **Figure 19-11**.
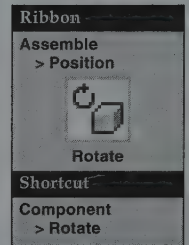
> **NOTE**
>
> Updating the assembly overrides a move or rotate conducted using the **Move Component** or **Rotate Component** tools if constraints are present, bringing components back to their original constrained location.

Figure 19-10.
Using the **Move Component** tool to move a constrained component.

Selected component

Moved
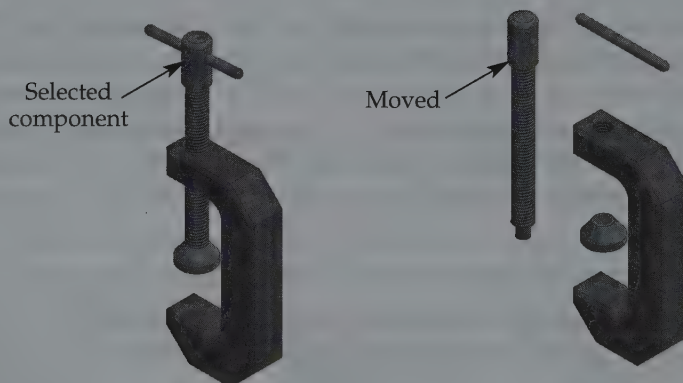
Figure 19-11.
Using the **Rotate Component** tool to rotate a constrained component.

Selected component

Rotated
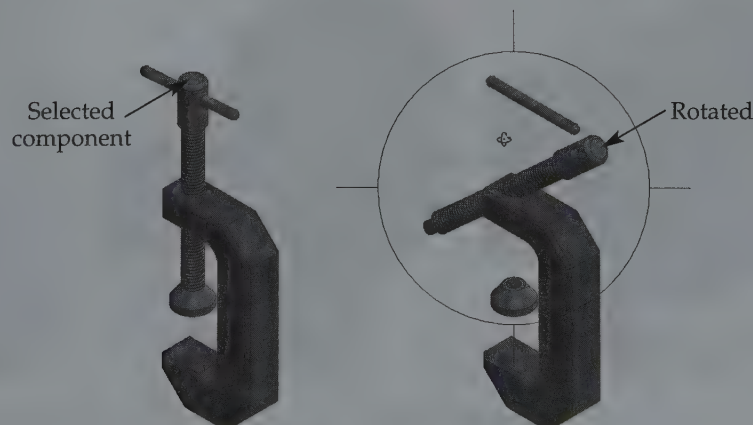
Changing Component Colors

Components initially appear in an assembly according to the color assigned to part models. Often the material and color properties are the same or similar for different components. Using the same colored components is appropriate for an assembly to resemble the final product. However, the result is an assembly with components that can be difficult to distinguish for modeling purposes. Overriding component color in the assembly is often necessary to make designs easier to understand and use. See **Figure 19-12**.

override: A temporary change to the current style or color settings.

To *override* the component color in an assembly, select the component in the browser or graphics window and choose a color from the **Color Override** flyout. See **Figure 19-12**. You can also specify a color override by right-clicking on a component and selecting **iProperties** to change the color using the **Color Style** drop-down list in the **Occurrence** tab of the **iProperties** dialog box. Color override in the assembly environment does not affect component color in the component file.

Exercise 19-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-5.

Sectioning Assemblies

If you have difficulty clearly observing a design, visualizing assembly component relationships, or accessing features when constraining or creating components, you may be able to solve the problem with an assembly section view. See **Figure 19-13**. When you finish using the section view, pick the **End Section View** button from the **Section View** flyout button to return to the complete, unsectioned assembly.

cutting plane: A plane that defines the location at which a component is sliced to show interior features.

Use section view tools to expose hidden component features. You can create different section views, depending on the application and the amount of material to remove, and then return the model to the unsectioned view when you are finished. To apply a section, you must select faces or planes to specify the *cutting plane*.

Figure 19-12.
Components in an assembly are easier to distinguish when they are different colors. A quick way to change component color is to use the **Color Override** flyout.

All Components Are
Same Color

All Components Are
Different Colors

Figure 19-13.
Knowing how to use section views will make it easier to create internal components in place.

Component

Section

First Component

Creating a Component in Place

Final Assembly
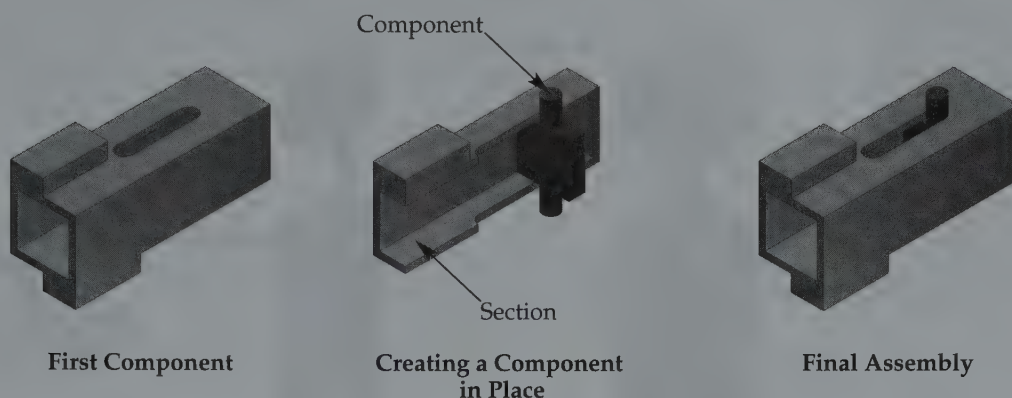
Quarter Section

Access the **Quarter Section View** tool to create a *quarter section view* by picking two intersecting faces or work planes. The side of the face or plane you select determines which quarter of material remains. See **Figure 19-14**. If the section looks acceptable, press [Esc], right-click and pick **Done**, pick anywhere in space, or access another tool to exit. If the section is not correct, you may be able to solve the problem by right-clicking and selecting **Flip Section** to change the direction of sight, or pick **Three Quarter Section View** to create a three-quarter section view.

Half Section

Access the **Half Section View** tool to create a *half section* by picking a face or plane. The side of the face or plane you select determines which half of the material remains. See **Figure 19-15**. If the section looks acceptable, press [Esc], right-click and pick **Done**, pick anywhere in space, or access another tool to exit. If the section is not correct, you may be able to solve the problem by right-clicking and selecting **Flip Section** to change the direction of sight.

Ribbon
View
> Appearance

Quarter Section View
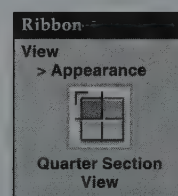
quarter section view: An assembly section view that that removes three-quarters of material at two intersecting faces or planes.

Ribbon
View
> Appearance
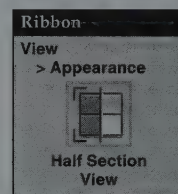
Half Section View

half section view: An assembly section view that that removes one-half of material at a face or plane.

Figure 19-14.
Creating a quarter section view.

Selected positions

Selected work planes

Original Assembly
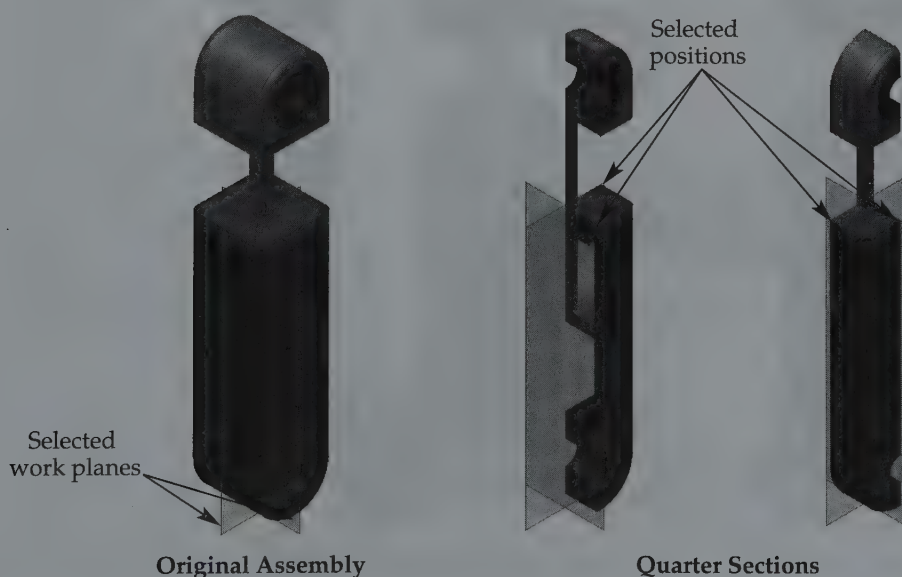
Quarter Sections

Figure 19-15.
Creating a half section view.

Selected work plane
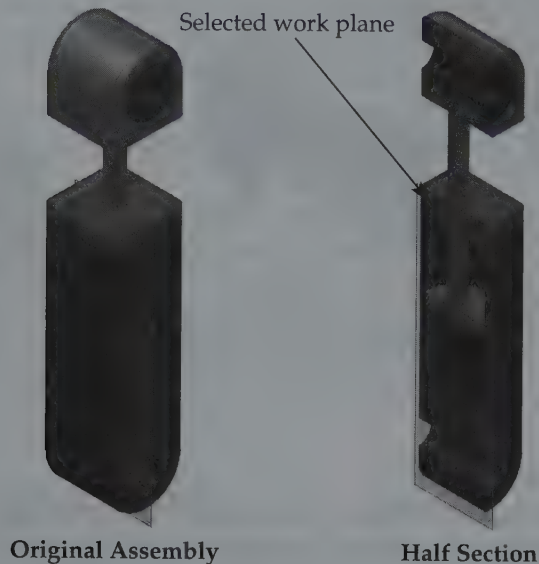
Original Assembly | Half Section

Three-Quarter Section

Access the **Three Quarter Section View** tool to create a *three-quarter section* by picking two intersecting faces or planes. The side of the face or plane you select determines which quarter of material is removed. See **Figure 19-16**. If the section looks acceptable, press [Esc], right-click and pick **Done**, pick anywhere in space, or access another tool to exit. If the section is not correct, you may be able to solve the problem by right-clicking and selecting **Flip Section** to change the direction of sight, or pick the **Quarter Section View** option to define a quarter section view.

Figure 19-16.
Creating a three-quarter section view.
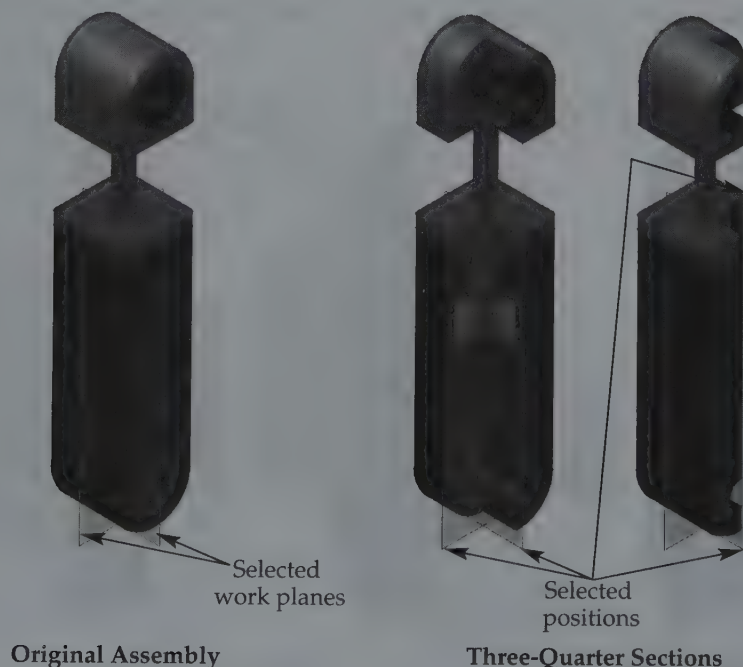
Selected
work planes

Selected
positions

Original Assembly | Three-Quarter Sections

Exercise 19-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-6.

Flexible Subassemblies

A subassembly is made of the same components and looks the same regardless of the parent assembly to which you add the subassembly. This becomes an issue when the subassembly must appear differently in different assemblies. For example, when you add the subassembly shown in **Figure 19-17** to an assembly, each instance of the subassembly looks the same. Create a *flexible subassembly* in order to display subassembly components in different conditions. **Figure 19-17** shows turning the subassembly adjustment knob for several instances in the same parent assembly file.

flexible subassembly: A subassembly you can adjust to display unique characteristics or kinematic conditions.

To make a subassembly flexible, right-click on the subassembly in the browser or graphics window and select **Flexible**. Once they become flexible, unconstrained subassembly components are free to move according to possible degrees of freedom and can be constrained as needed. To return a subassembly to its original orientation, as defined in the subassembly file, deselect the **Flexible** menu option.

Figure 19-17.
Multiple instances of a subassembly are initially positioned identically, but you can change the position of each to show an alternative display or convey kinetic conditions. (Model courtesy of Ethan Collins)

Subassembly

Assembly with Six Subassemblies Positioned Identically

Alternative Display of the Assembly

Inventor provides design view, positional, and level-of-detail representations to aid in working with assemblies, especially large complex assemblies. Use representations to access a specific assembly display configuration, without spending time recreating the appearance when needed. Access and manage representations using the **Representations** folder in the browser, or using specific representation tools.

Design View Representations

design view representation: An assembly view that displays components in an alternative display format.

Ribbon

View
> **Saved Views**

Design View Representations

A *design view representation* "takes a picture" of an assembly with a specified set of component colors and styles, component selection, visibility status, and flexibility condition. For example, you can create design view representations with different colored components, a section view of the assembly, or certain components disabled or turned off. You can then recall a design view representation as needed, without reassigning the various view characteristics.

Select the **View** node from the **Representations** folder to control design view representations in the browser, or manage design view representations using the **Design View Representations** dialog box. See **Figure 19-18**. The Master view represents the unaltered assembly. You cannot modify the Master view, such as by overriding component colors, and you cannot delete it. You can use the Default view to develop a design view representation that is different from the Master view, or create new views as needed.

To create a new view, right-click on the **View** node and select **New**, or type a name in the **Design View Representations** text box and pick the **New** button. Specify a new view name, or edit an existing name to something more descriptive, such as Half section or Bracket-off. Then, to develop a view, make changes to the Default view by assigning color overrides, disabling components, turning off component visibility, adding section views, or adjusting other display characteristics as needed, depending on the purpose of the view.

Changes made to the active view are automatically saved with the view in the current assembly file. To activate a different design view representation from the browser, double-click on the view, select the view from the **Design View Representations** flyout, or right-click on a view and select **Activate.** From the **Design View Representations** dialog box, activate a view by picking the view and selecting the **Active** button. The active design view displays the view characteristics assigned to the view. To delete a design

Figure 19-18.
A—Expand the **Views** node in the **Representations** folder of the browser to control design view representations. B—Use the **Design View Representations** dialog box to control design view representations

Enter a new view name

A   B
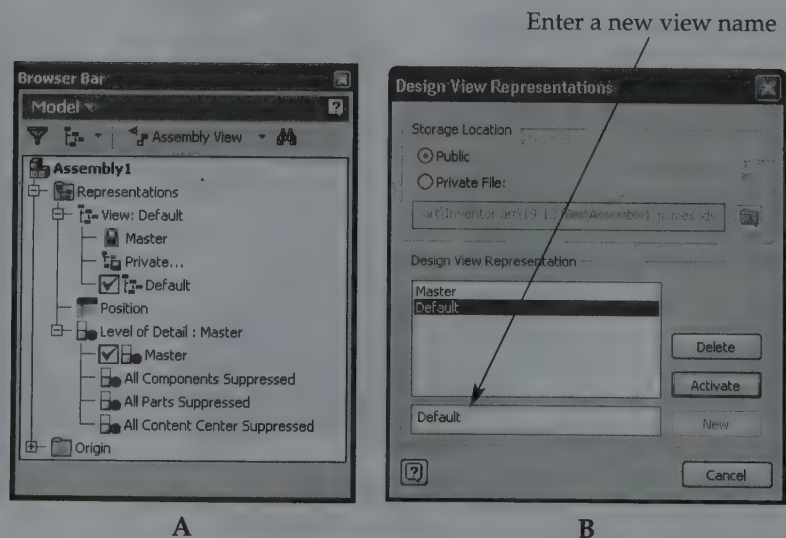
view representation from the browser, right-click on a view and select **Delete**, or while using the **Design View Representations** dialog box, pick the view and select the **Delete** button.

> **NOTE**
>
> The active **Public** radio button in the **Design View Representations** dialog box indicates that views are saved to the assembly file, which is appropriate from most applications. The Private... item in the browser and **Private File** options in the **Design View Representations** dialog box allow you to control the location of a separate design view representation .idv file, used by older versions of Inventor.

Exercise 19-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-7.

Positional Representations

The main purpose of a *positional representation* is to override specific assembly constraints without modifying or eliminating the original assembly constraints. Positional representations are also required for generating overlay drawing views. **Figure 19-19** shows an example of using positional representations. In this example, Master is the fully constrained and nonextended assembly with a 0 offset mate constraint; HALF uses a mate constraint override of 1.625"; and FULL uses a mate constraint override of 3.25".

Select the **Position** node from the **Representations** folder in the browser to control positional representations. The Master view represents the unaltered assembly and appears when you create a new positional representation. To create a positional representation, right-click on the **Position** node and select **New**. Most assembly tools become disabled and a new positional representation named Position1 appears, along with the

> **positional representation:** An assembly view that displays the components in a specific position.

Figure 19-19.
Master is the default positional representation, but you can create others, such as the HALF and FULL positional representations used to illustrate various piston extensions.
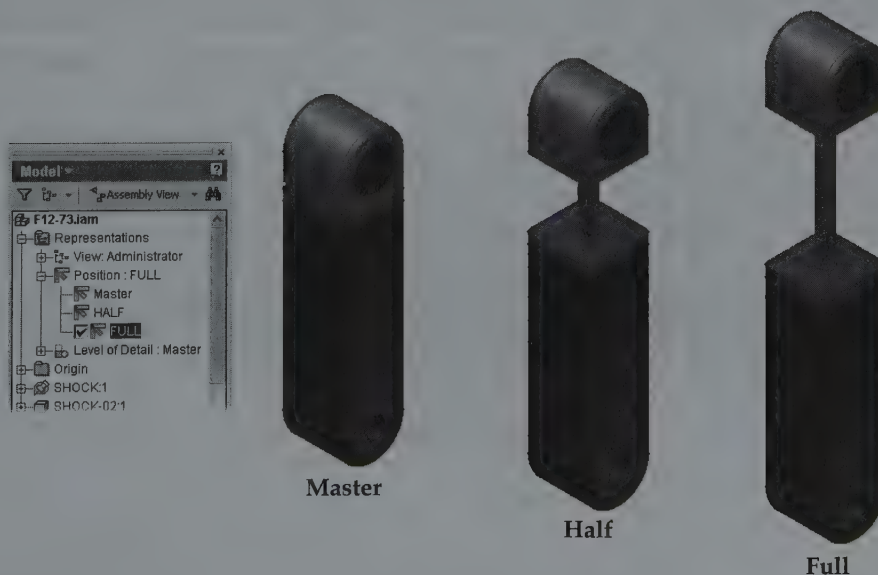
Master

Half

Full

Master view. Specify a more descriptive name if necessary, such as HALF or FULL, as shown in **Figure 19-19**.

Then, to develop a positional representation, override component placement and constraints. This process is similar to dragging, modifying, or driving constraints and components, but you use the **Override Object** dialog box. To access the **Override Object** dialog box, shown in **Figure 19-20**, right-click on a component, rectangular or circular component pattern, or constraint to override and select **Override**. You can also double-click on a constraint. The options in the **Override Object** dialog box are specific to the item you override. Use the **Positional Representation** drop-down list to select the positional representation in which the override occurs. Select the **All** option to apply the override to all positional representations except for Master.

The options on the **Constraint** tab, shown in **Figure 19-20A**, are available when you override a constraint. To override constraint suppression, pick the **Suppression** check box to suppress an enabled constraint or **Enable** from the drop-down list to enable a suppressed constraint. To override the value of a constraint, select the **Value** check box, and then enter a different value in the text box.

The options on the **Pattern** tab, shown in **Figure 19-20B**, are available when you override a rectangular or circular component pattern. Override rectangular pattern row and/or column spacing by picking the appropriate check boxes. Then specify a value using the corresponding text boxes. Override a circular pattern angle by specifying a value in the text box. Pattern values override spacing between pattern copies, even though Inventor identifies the value as *offset*.

The options on the **Component** tab, shown in **Figure 19-20C**, are available when you override a component. To override component grounding, pick the **Grounding** check box. To override the component location, or offset, pick the **Position Offset** check box, and then drag the component to a new location in the graphics window. Pick the **Positional Representation** check box to override the positional representation selected from the drop-down list. If the component is a subassembly, pick the **Flexible Status** check box to override the flexibility of the subassembly by selecting **Flexible** or **Non-Flexible** from the drop-down list.

Figure 19-20.
A—The **Constraint** tab of the **Override Object** dialog box. B—The **Pattern** tab of the **Override Object** dialog box. C—The **Component** tab of the **Override Object** dialog box.

A   B   C
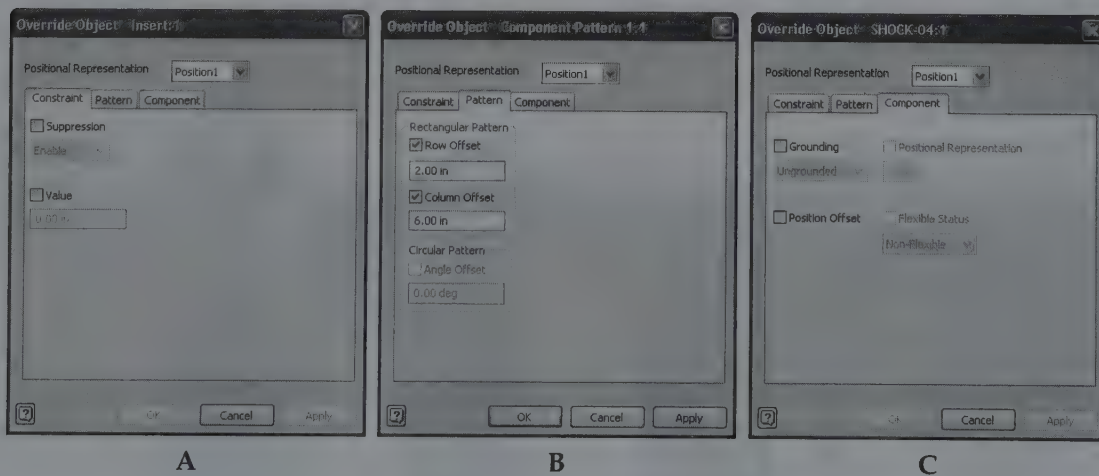
Tools such as **Move Component**, **Rotate Component**, and the various section view options are available when you create a positional representation. However, the main purpose of establishing a positional representation is to use overrides.

Changes made to the active view are automatically saved with the view in the current assembly file. To activate a different positional view representation, double-click on the view or right-click on a view and select **Activate**. The active design view displays the overrides and positional characteristics assigned to the view. To delete a positional view representation, right-click on a view and select **Delete**.

If you fully understand how to edit components and drive constraints, as described earlier in this chapter, you are better prepared to add positional representations.
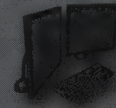
Exercise 19-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 19-8.

Level-of-Detail Representations

A *level-of-detail representation* controls the number of components suppressed in the assembly, and provides a way to work with specific components without the clutter of the entire parent assembly. **Figure 19-21** shows an example of using level-of-detail representations to recall specific groups of unsuppressed components.

> **level-of-detail representation:** An assembly view in which a set of components are suppressed.

Select the **Level of Detail** node from the **Representations** folder in the browser to control level of detail representations. The Master view represents the unaltered assembly. To suppress all components in the assembly, double-click on the All Components Suppressed view. Use this option before saving and closing a file to decrease load time the next time you open the file, or to suppress all components and then individually unsuppress certain components. Double-click the All Parts Suppressed view to increase performance in large assemblies and work only with subassemblies. Double-click the All Content Center Suppressed view to increase performance in large assemblies by suppressing all content center components.

To create a level-of-detail representation, right-click on the **Level of Detail** node and select **New Level of Detail**. Specify a more descriptive name if necessary, such as those shown in **Figure 19-21**. Then suppress components as appropriate and save the file to save the changes made to the level-of-detail representation. You can now work between different levels of detail views to suppress and unsuppress groups of components.

Use Master design view, positional, and level-of-detail representations to return to the standard assembly display, and as a point of beginning when creating new representations.

Figure 19-21.
An example of a
guitar assembly
with three levels of
detail for working
on specific portions
of the assembly
and increasing
model performance.
(Model courtesy of
Ethan Collins)

Ribbon

**Manage
> Update**

Defer Update
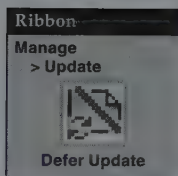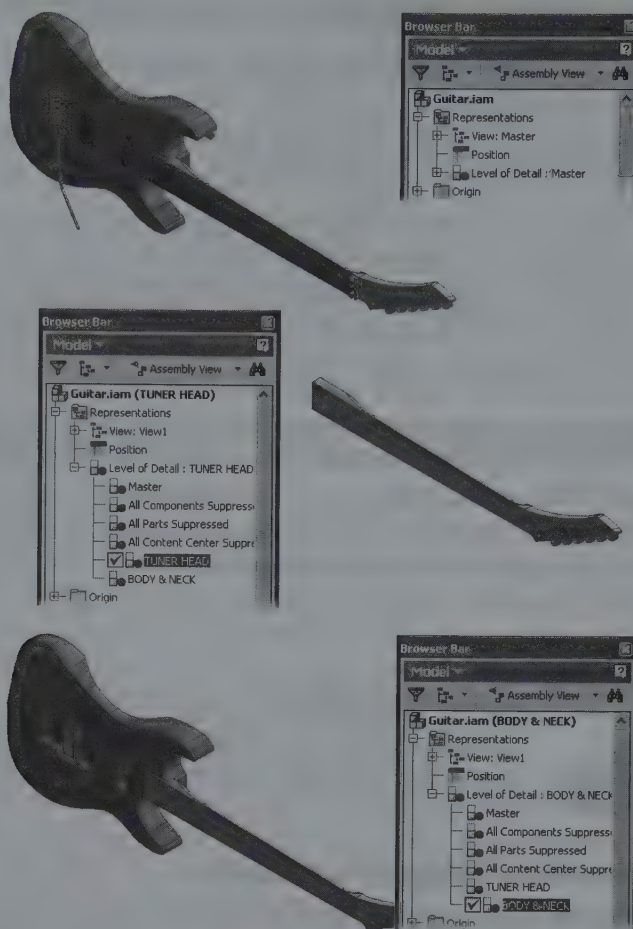
NOTE

Large assemblies with many components can often be cumbersome to work with because large assembly files reduce performance and consume significant memory. One option for increasing performance is to postpone updates of changes made to components until you update the file. Use the **Defer Update** tool to postpone updates.

Chapter Test

1. Briefly describe a component pattern.
2. Describe and give an example of an associative pattern.
3. What are mirrored components?
4. What is the function of the **Replace Component** tool?
5. Briefly explain how to override the component colors in an assembly without affecting the color in the component file.
6. Describe a quarter section view.
7. What is a half section view?
8. What is a flexible subassembly?
9. Explain the basic function of a design view representation.
10. What is a level-of-detail representation?

Problems

1–2 Instructions: Follow the specific instructions for the following problems.

1. Ensure that the Exercises and Problems project is active. Access the **Open** dialog box and pick the **Samples** library to navigate to the **Samples** folder. Open the Personal Computer.iam file from the following folder: Samples/Models/Assemblies/ Personal Computer. Use view tools and the browser to explore the model. Activate the Ribbon Cable Creation level-of-detail representation and then return to the Master view. Close the assembly file without saving.

2. Ensure that the Exercises and Problems project is active. Access the **Open** dialog box and pick the **Samples** library to navigate to the **Samples** folder. Open the Suspension.iam file from the following folder: Samples/Models/Assemblies/ Suspension. Use view tools and the browser to explore the model. Activate the Right Bump positional representation and then return to the Master view. Close the assembly file without saving.

3–6 Instructions:

- *Prepare new components for insert or build components in-place as appropriate to create the following assemblies.*
- *Use appropriate names and properties for all components.*
- *Use appropriate sketching techniques and fully constrain sketch geometry.*
- *Add all necessary assembly constraints.*
- *Use your own judgment and approximate dimensions when necessary.*
- *Add as much information as possible to the **iProperties** dialog box for component and assembly files. Assign the specified material and color to parts.*
- *Follow the specific instructions for each problem to create the assemblies.*

Note: Inventor dimensional constraint appearance may not comply with ASME standards.

3. **Title:** SHOCK ABSORBER
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-106-001-01
   Project: UTILITY VEHICLE
   Save as: P19-4.iam
   Specific Instructions: Design a shock absorber assembly similar to the example shown.

Assembly

Individual Parts
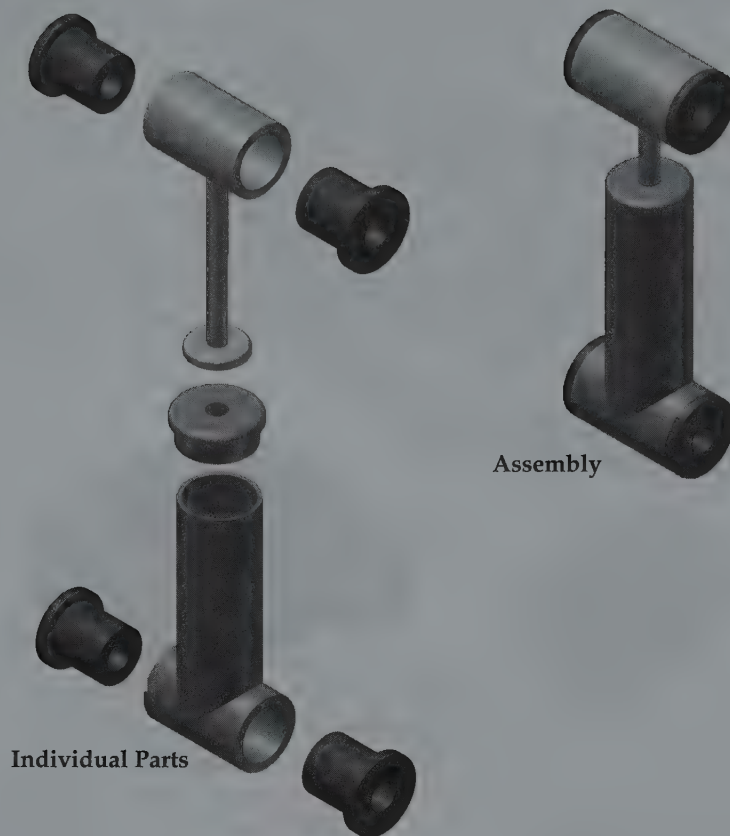
4. **Title:** SHOCK ABSORB SYSTEM
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-106-001
   Project: UTILITY VEHICLE
   Save as: P19-5.iam
   Specific Instructions: Design a shock absorption system similar to the example shown. Insert two copies of the shock absorber from Problem 19-3. Create a sheet metal bracket as shown and insert a second copy.

Subassemblies   **Parts**   **Assembly**
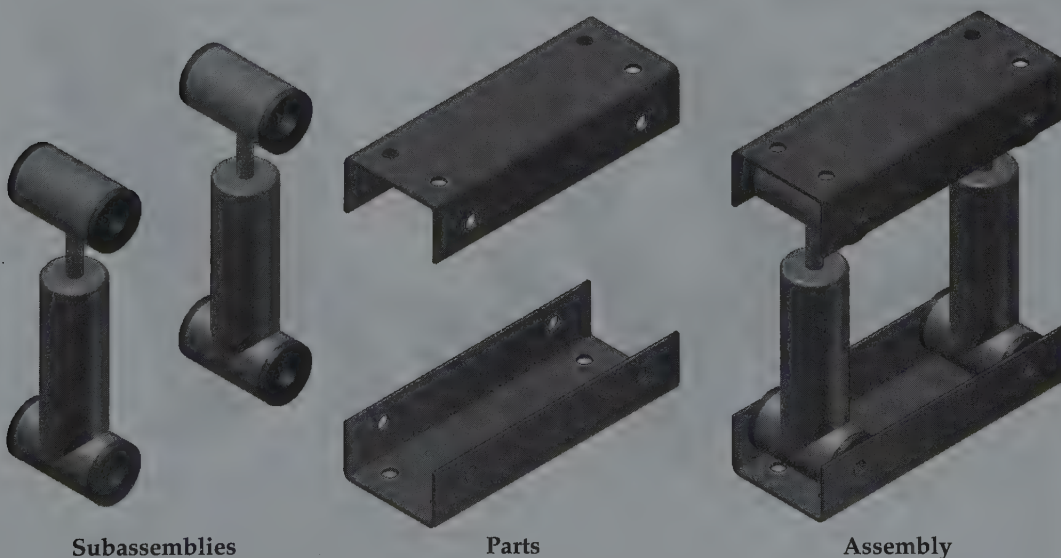
5. **Title:** DISPENSER
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-107
   Project: DISPENSER
   Save as: P19-6.iam
   Specific Instructions: Design a dispenser assembly similar to the example shown. Create the subassembly shown using a separate pin, two bushings, and a wheel file. Then insert the subassembly into a file with the base part to create the assembly. Use adaptive modeling and then change the dimensions to explore correct adaptivity.
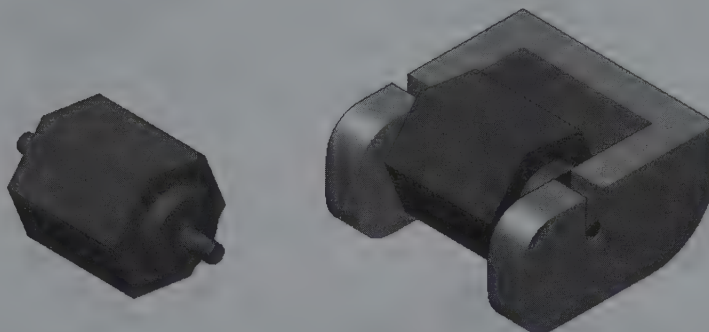
Subassembly   Assembly

6. **Title:** LEVER SUBASSEMBLY
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-108
   Project: LEVER
   Save as: P19-7.iam
   Specific Instructions: Design a lever assembly similar to the example shown. Use adaptive modeling and then change the dimensions to explore correct adaptivity.
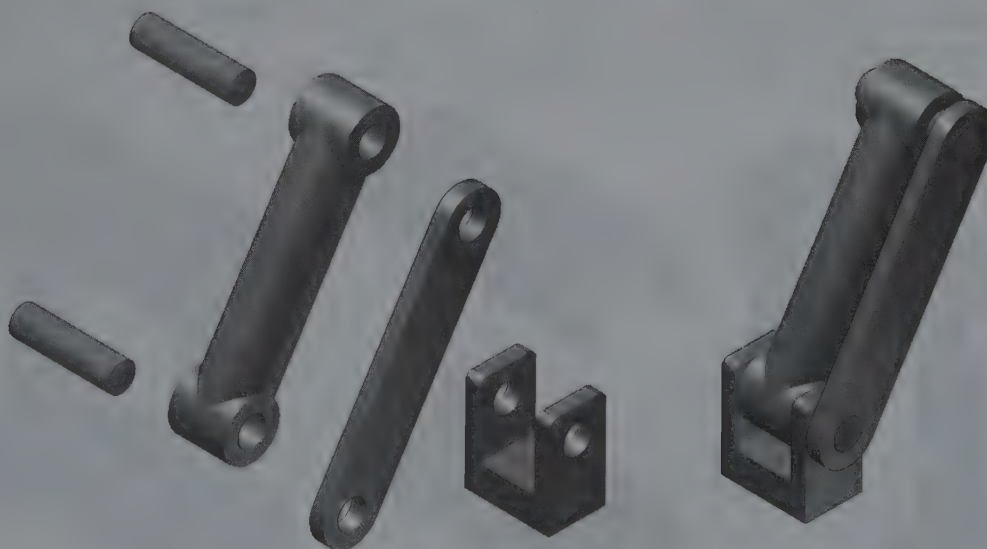
Parts   Assembly
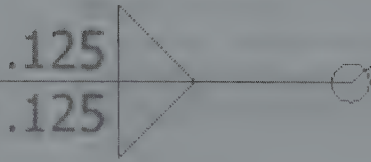
Weldments 20

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Begin new weldments and convert assemblies to weldments.
✓ Prepare weldment components for welding.
✓ Place fillet, groove, and cosmetic welds.
✓ Add welding symbols and end fills.
✓ Machine and edit weldments.

Inventor includes *welding* tools that allow you to model a *weldment*. See **Figure 20-1**. A weldment model is an assembly model that allows you to apply pre-weld, weld, and post-weld operations to replicate developing a welded product. This chapter explains the process of creating a weldment model.

welding: The process of joining pieces of like materials, usually metal or plastic, by heating the material to a temperature that is high enough to cause softening or melting, so the pieces combine grain structure and become joined together.

weldment: An assembly in which components are fixed together with welds.

Figure 20-1.
An example of a weldment model created by preparing and assembling components, and then welding the components together. Look closely to see the weld bead representations.
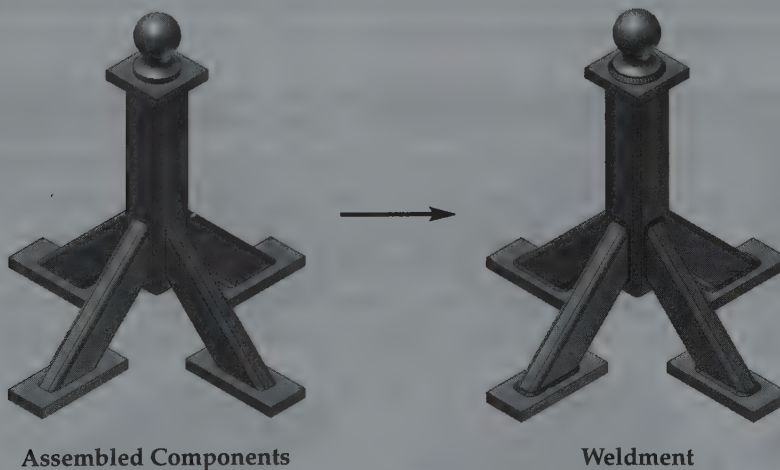
Assembled Components   Weldment

The initial phase of weldment modeling is much like creating an assembly. Start with a weldment model template and insert existing components or create components in place. Then constrain the components to form the assembly. The next phases are specific to weldment modeling. Add pre-weld, or preparation, features to prepare the model for welding. Then weld components together using weld features. Finally, if necessary, apply post-weld, or machined, features to finish the weldment. The standard approach to weldment modeling varies depending on the application. For example, you may want to prepare components before assembly with constraints. You also have the option of converting an assembly to a weldment.

Weldment Templates

Assembly and weldment files carry the file extension .iam. Typically, when you begin a new design, you specify whether the assembly is for a non-weldment assembly or a weldment application. Use a weldment template to create a weldment using weldment tools and options.

Converting Assemblies to Weldments

If you begin modeling from a non-weldment assembly template, you must convert the assembly to the weldment assembly environment to access weldment tools. Access the **Convert to Weldment** tool to change from an assembly design to a weldment design. Once you transition from an assembly to a weldment, you cannot return to an assembly application. An alert advises you of this rule and asks if you want to proceed. If necessary, save a copy of the assembly before converting it to a weldment. Select the **Yes** button to continue the conversion and display the **Convert to Weldment** dialog box.

Select a radio button from the **Standard** area to set the standard to be used when applying welds and welding symbols. Select a weld bead material corresponding to the product material from the **Weld Bead Material** drop-down list. Select an option from the **Default BOM Structure** drop-down list to adjust the status of the weldment in the assembly bill of material. The **Inseparable** option is suitable for most weldment applications. Options in the **Feature Conversion** area are enabled if the assembly includes features. Pick the **Weld preparations** radio button to convert assembly features to weldment preparations, or pick the **Machining features** radio button to convert assembly features to weldment-machined features. Pick the **OK** button to finalize the conversion and enter the weldment work environment.

> **CAUTION**
>
> The **Weldment** environment includes modeling tools, such as **Extrude**, specifically for weldment model applications, including preparing and machining components. Do not confuse these feature tools with those used to create part or subassembly models, even though they function the same. Using feature tools to modify a weldment does not affect component models outside of the weldment file.

Exercise 20-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-1.

Ribbon

Assemble
> Convert

Convert to Weldment

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-2.

Material Preparation

Material preparation allows you to form a joint, groove, *root opening*, hole, or other feature required to achieve the proper weld. The type, location, and size of preparation features are specific to the welding application. **Figure 20-2** shows examples of common weld preparations. Although you can prepare components for welding in component files, material preparation tools in the weldment work environment mimic the process of preparing components for welding. You can add preparations before or after constraining components.

The **Preparations** node in the browser stores preparations. Activate the preparation mode from the ribbon, or double-click on the **Preparations** node in the browser or right-click on the node and select **Edit**. The preparation work environment provides common feature, work feature, feature pattern, and parameter tools appropriate for pre-weld operations. These tools behave the same as those in the standard part model environment, except that you can only remove material when adding preparations.

Use the **Extrude, Revolve**, and **Sweep** tools to cut a sketch profile for applications such as J grooves, U grooves, and slots for slot welds. Use the **Hole** tool to add holes for spot welds. **Figure 20-3** shows examples of sketched features and holes added as preparations. Use the **Fillet** tool to create J grooves and the **Chamfer** tool to produce bevel and V grooves, as shown in **Figure 20-4**. Use the **Move Face** tool to move faces for a root opening. Pattern features such as holes and slots using the **Mirror, Rectangular Pattern**, and **Circular Pattern** tools.

When you are finished preparing material, exit the preparation mode by double-clicking on the weldment icon in the browser, or activating another weldment process. You can also right-click on the **Preparations** node and select **Finish Edit**, or use the **Return** tool. Re-enter the preparation mode as needed throughout the weldment design process.

material preparation: All tasks required and performed before welding.

root opening: A space between components used to achieve a proper weld.

Ribbon

Weld
> Process
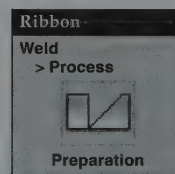
Preparation

Figure 20-2.
Examples of weld preparations.

V-groove with a root opening

Bevel groove

Spot-weld holes

Slot-weld slot
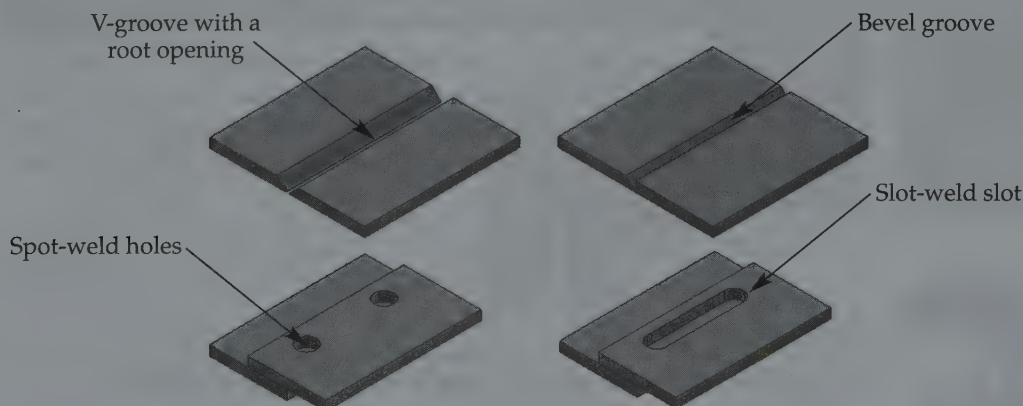
Figure 20-3.
Examples of weld preparations created using extruded sketch profiles and holes.

.125
.016
.031

.125
.016
.031

J Groove with Root Opening

U Groove

.250
.500
.250
.500

.125
.500

Holes for Spot Welds

Slot for Slot Weld
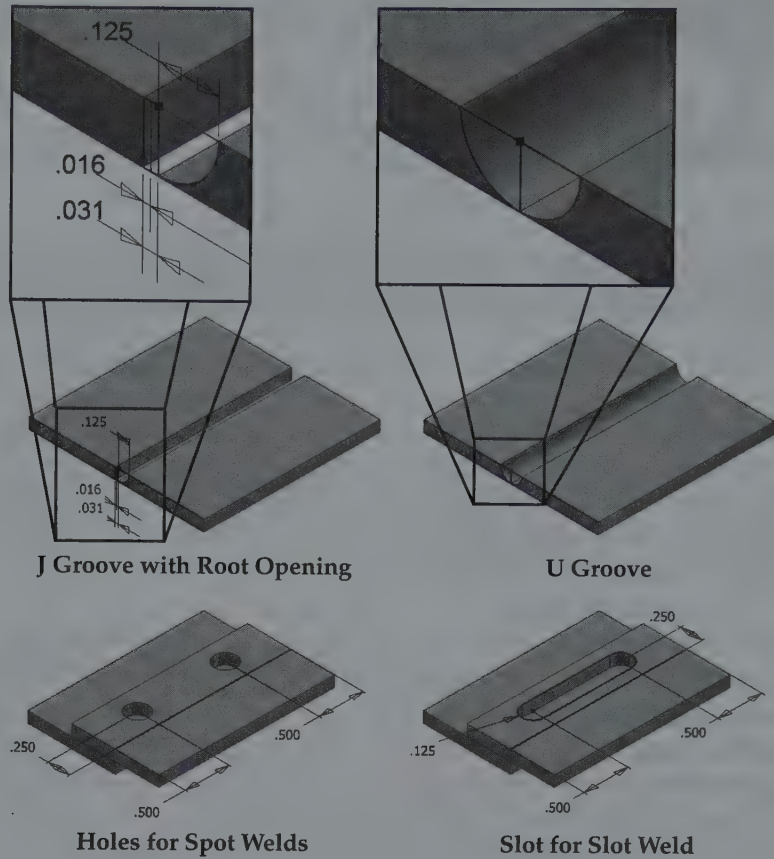
Figure 20-4.
Examples of weld preparations created using the **Chamfer** tool.

Bevel Groove

V Groove
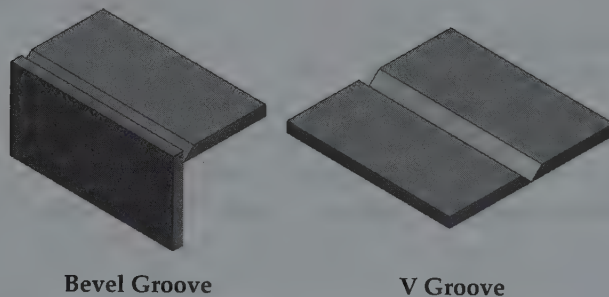
PROFESSIONAL TIP

In some situations, you can use assembly constrains to mimic a preparation without creating a preparation feature. For example, you can use an offset mate constraint to define a root opening. However, you must compensate for the root opening when designing the component.

Exercise 20-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-3.

Welding

Weld placement, or *welding*, is the next phase of weldment modeling. Weld placement often fills the joint, groove, or hole created during material preparation. Traditionally, weld specifications appear on a drawing, pointing to the weld joint. Inventor allows you to add welds to the model, much like creating an actual weldment. You can then reference the weld information when preparing a drawing.

The **Welds** node in the browser stores welds. Activate the welding mode from the ribbon, double-click on the **Welds** node in the browser, or right-click on the node and select **Edit.** The welding work environment provides welding, work feature, and parameter tools. The work feature and parameter tools behave the same as those in the standard part model environment.

Figure 20-5 shows a weldment with examples of the fillet, groove, and cosmetic welds you can create using welding tools. Use the **Fillet Weld** tool to create an authentic fillet weld. Use the **Groove Weld** tool to create non-fillet welds, such as spot, slot, V-groove, and butt welds. Fillet and groove welds display and calculate weld bead mass. Weld beads can include a .bmp image of weld geometry, if you select an appropriate weld bead material and color. Use the **Cosmetic Weld** tool to locate and specify any type of weld. A cosmetic weld highlights the joint without creating a bead representation, and includes weld bead cross-sectional area properties. All welds can include a *welding symbol* that you can later recall when creating a weldment drawing.

The **Bead** folder in the **Welds** node of the browser stores weld beads. Welding symbols are listed separately in the browser, and consume a copy of the model weld bead, although the weld bead is also displayed in the **Beads** folder. Before you begin welding, you may want to select or confirm weld properties, including material and color. To specify weld properties, right-click on the **Welds** node in the browser, or any weld in the graphics window, and select **iProperties** to access the **iProperties** dialog box. Here you can adjust options specific to welds not found in the **iProperties** dialog box of the file. Pick the appropriate weld bead material from the **Material** drop-down list in the **Physical** tab.

When you are finished welding, exit the welding mode by double-clicking on the weldment icon in the browser or activating another weldment process. You can also right-click on the **Welds** node and select **Finish Edit,** or use the **Return** tool. Re-enter the welding mode as needed throughout the weldment design process.

weld placement (welding): The process of specifying the type, size, and location of weld applied to assembly components.

Ribbon
Weld
> Process

Welds

welding symbol: A symbol that indicates welding specifications.

Figure 20-5.
A weldment with examples of fillet, groove, and cosmetic welds.

Cosmetic welds

Groove welds

Fillet welds
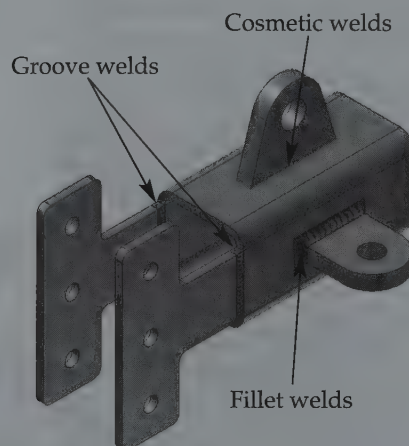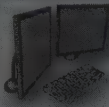
Adjust weld bead properties in separate weldment templates for specific applications. For example, create a mild steel weldment template with the corresponding weld bead material for preparing mild steel weldments.

Fillet Welds

Access the **Fillet Weld** tool to create true *fillet welds* using the **Fillet Weld** dialog box. See **Figure 20-6**. You can also use the **Fillet Weld** tool to place other welds, such as plug, spot, and combined groove and fillet welds. Use the **Select Face(s) 1** button to pick the first face(s) to weld. Next, pick the **Select Face(s) 2** button and select the second face(s) to weld. When picking multiple tangent faces, choose the **Chain** check box so you do not have to select the faces individually. The combination of selections locates the joint as shown in **Figure 20-6**.

Ribbon

Weld
> Weld

Fillet

Type

[W]

To deselect faces, with the **Select Face(s) 1** or **Select Face(s) 2** still active, hold down [Shift] or [Ctrl] and pick the faces to remove from the selection.

When welding all-around components, you must pick multiple adjacent faces for each side.

Figure 20-6.
Using the **Fillet Weld** dialog box to create a fillet weld.
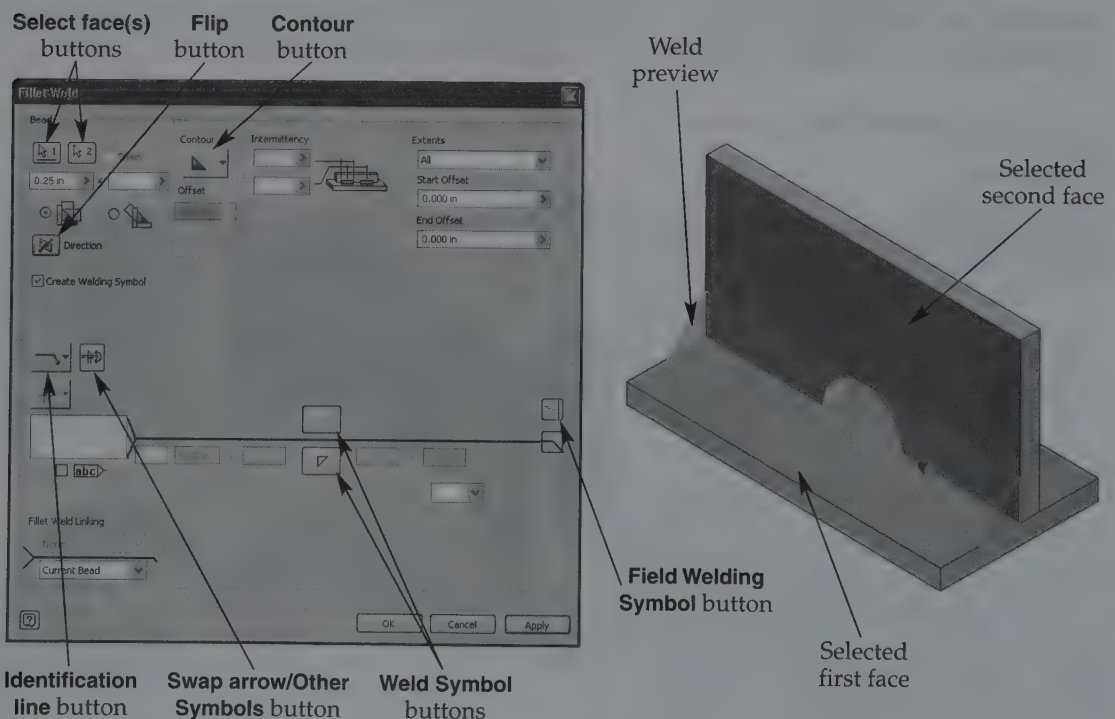
The options below the face selection buttons control the weld size. Pick the **Leg Length Measurement** radio button to specify the weld bead size according to *leg* length. Use the **Leg 1** text box to specify the length of the first leg, which corresponds to the first selected face. Then, only if you want to create a different size second leg, specify a length using the **Leg 2** text box, which corresponds to the second face you selected. Select the **Throat Measurement** radio button to identify the weld bead size according to the *throat* using the **Throat Measurement** text box.

Select the appropriate *contour* using the **Contour** flyout. When you select the **Convex** or **Concave** option, the **Offset** text box activates, allowing you to define the distance from the imaginary line connecting the fillet weld *toes* to the weld bead face. The larger the convex or concave offset, the smaller the weld bead contour radius. The **Offset** text box does not apply to a **Flat** contour because the actual face of the weld bead connects the weld bead toes in a flat contour.

The **Intermittency** area establishes whether the weld bead occurs along the entire length of a joint or according to a specified *weld length* and *pitch*. For example, the weld shown in **Figure 20-7** has a weld length of 25 mm and a pitch of 45 mm, which means there is a 20 mm space between weld beads. If both text boxes are clear, the weld bead occurs along the entire joint. To create an intermittent weld bead, specify the weld length using the **Length** text box, and specify the weld pitch using the **Pitch** text box.

The **Extents** area allows you to define the extents of the weld. See **Figure 20-8**. Pick the **All** option to weld along the entire joint. Pick the **From-To** option to weld between two parallel faces or planes selected using the **From** and **To** buttons. Pick the **Start-Length** option to begin the weld at a distance specified in the **Start Offset** text box from the start of the joint, and using the weld length specified in the **Length** text box. Use the **Flip Direction** button on the left side of the dialog box to reverse the start point.

Adding a Welding Symbol and Finalizing the Weld

Pick the **Create Welding Symbol** check box to create a welding symbol for reference in the model and to extract to a weldment drawing. The **Fillet Weld Linking** area defines the *arrow side* and *other side* welding symbol specifications according to the model weld bead settings previously described. The default **None** option allows you to create a welding symbol without referencing the model weld bead. To use the parameters specified in the **Bead** area of the **Fillet Weld** dialog box, pick the **Current Bead** option from the drop-down lists. Notice that several welding symbol options become unavailable, because you have already set the data using options in the **Bead** area.

leg: The distance from the intersection of the two selected faces, or joint, to the intersection of the weld bead and one of the component faces.

throat: The shortest distance from the intersection of the two selected faces, or joint, to the weld bead face.

contour: The external shape of the weld surface, such as flat, concave, or convex.

toes: The intersection of the weld bead and one of the component faces.

weld length: The length of each weld.

pitch: The center-to-center distance between weld lengths.

arrow side: The side of the joint where the weld occurs and the welding symbol points.

other side: The opposite side of the joint from the arrow side, where a weld may occur, but the welding symbol does not point.

Figure 20-7.
You must know the weld length and pitch to build an intermittent fillet weld.

20 mm
Space between weld beads
Weld length
25 mm
45 mm
Pitch
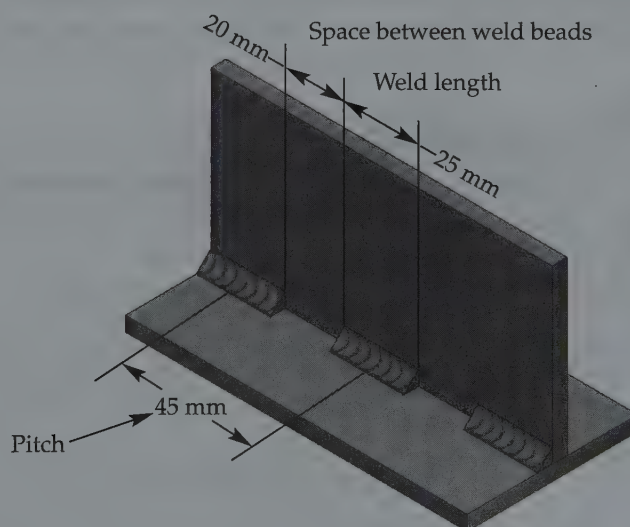
Figure 20-8.
An example of a cosmetic weld created using the **All** extents option and the **From-To** extents option.
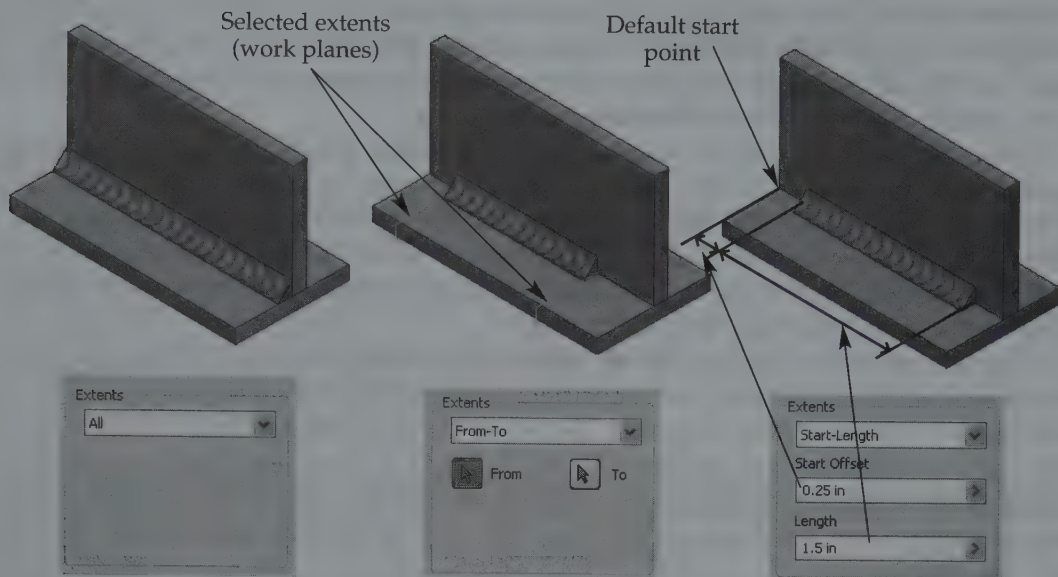
NOTE

The **Other Side** drop-down list of the **Fillet Weld Linking** area is available only if you specify an other side weld symbol using the **Other Side** button.

PROFESSIONAL TIP

Use fillet weld linking when adding a welding symbol to fillet welds so that the symbol becomes associated with the model weld beads.

reference line:
A horizontal line, attached to a leader pointing to a joint, that includes weld specifications.

weld symbol: The symbol added to the reference line that indicates the type of weld.

The additional options in the **Create Welding Symbol** area provide an interactive means of constructing a welding symbol on the *reference line*. **Figure 20-9** shows and describes basic options for developing a fillet welding symbol. Items in the **Create Welding Symbol** area are adjusted when you select or deselect a specific *weld symbol* and define welding symbol characteristics. Pick the **Apply** button to create the weld and remain in the tool, or pick the **OK** button to create the weld and exit.

Exercise 20-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-4.

Figure 20-9.
Basic options available for creating a fillet welding symbol. The options available vary
significantly, depending on the selected arrow side and other side weld symbol.

Item	Description
Identification Line	Controls the use and placement of the *identification line*. The **No Identification Line** option is the default and is appropriate for ANSI/AWS standards.
Swap Arrow/Other Symbols	Reverses the arrow side and the other side weld symbols.
Stagger	Available if you specify both arrow side and other side symbols. Defines the orientation of weld symbols. The **No Stagger** option is the default.
FILLET LEG +1/8 TOLERANCE -0	Add reference information with the tail.
☐ abc▷	Check to add a box around the reference note.
Prefix 1/4	Add a prefix before the weld size.
Leg 1 1/4 × 5/16	Specify the length of the first leg of the weld bead.
Leg 2 1/4 × 5/16	Specify the length of the second leg of the weld bead if it is different from the first.
Arrow Side Symbol	Select an arrow side weld symbol from the flyout. The welding symbol interface changes when you select a different symbol.
Length 2 - 5	Enter the length of an intermittent weld.
Pitch 2 - 5	Enter the pitch of an intermittent weld.
Contour G ▾ — ▾	Select a contour, such as the flat contour shown above, and a means of achieving the contour, such as the G selected to indicate grinding.
Other Side Symbol	Select an other side weld symbol from the flyout. The welding symbol interface changes when you select a different symbol.
Field Weld	Adds a *field weld* symbol.
All Around	Adds a *weld all around* symbol.

identification line: The line placed in addition to the reference line when using British welding drafting standards.

field weld: A welding process performed on the job site or in the field.

weld all around: A weld that continues around the entire feature at the designated location.

Groove Welds

groove weld: A weld bead typically applied to the groove created at the intersection of two, often parallel, components.

Access the **Groove Weld** tool to create common *groove welds,* such as square, double-V, and flare-bevel groove welds, using the **Groove Weld** dialog box. See **Figure 20-10**. You can also use the **Groove Weld** tool to add the groove weld portion of a combined groove and fillet weld. Use the **Select Face(s) 1** button to pick the first face(s) to weld. Next, pick the **Select Face(s) 2** button and select the second face(s) to weld. When picking multiple tangent faces, choose the **Chain** check box so you do not have to select faces individually. The combination of selections specifies the groove filled by the weld bead, as shown in **Figure 20-10**. You have the option of selecting the **Full Face Weld** check boxes as needed to define the length of the weld bead in reference to the longest face. See **Figure 20-11**.

> **NOTE**
>
> To deselect faces, with the **Select Face(s) 1** or **Select Face(s) 2** still active, hold down [Shift] or [Ctrl] and pick the faces to remove from the selection.

Figure 20-10.
Using the **Groove Weld** dialog box to create a square groove weld.
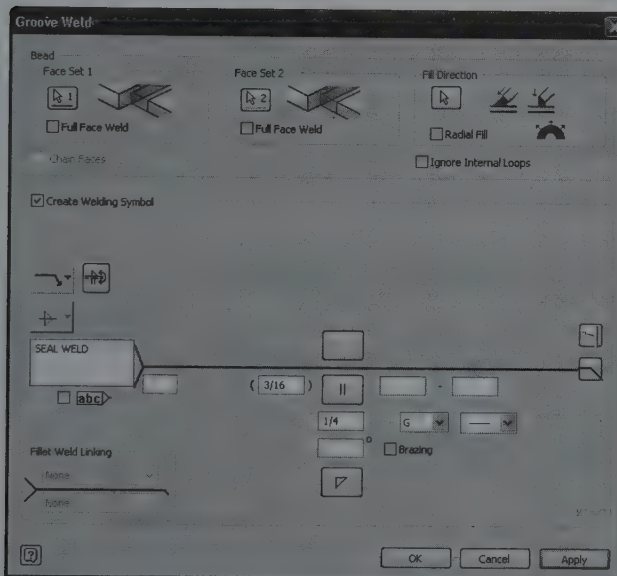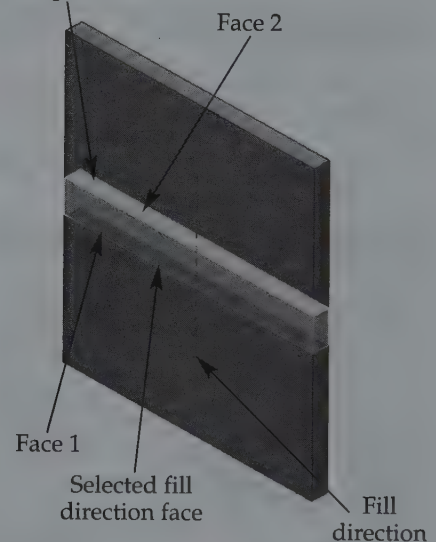
Figure 20-11.
These groove welds illustrate the effects of using the **Full Face Weld** option.
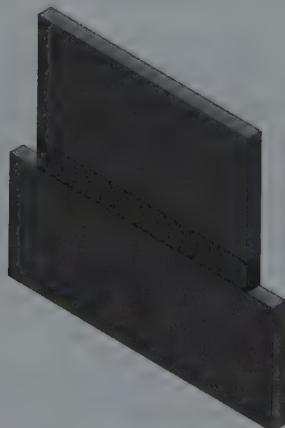
Without Full Face Weld Option   **With Full Face Weld Option**

Next, specify the *fill direction*. The fill direction is calculated automatically if you pick the **Full Face Weld** check boxes in the **Face Set 1** and **Face Set 2** areas. Otherwise, you must choose the **Select Fill Direction** button and pick an appropriate face, work plane, edge, axis, or two points associated with the selected component faces, depending on the required weld fill direction. Often, the fill direction is perpendicular to the selected component faces. For example, select the face shown in **Figure 20-10** to create the square groove weld. **Figure 20-12** shows examples of a groove weld created by choosing the same face sets, but different fill directions. You also have the option of selecting the **Radial Fill** check box to bypass the fill direction selection when welding around circular faces. **See Figure 20-13.**

If you select the **Ignore Internal Loops** check box, Inventor adds weld bead geometry to the entire face defined by an internal loop. If you do not select the **Ignore Internal Loops** check box, internal loops are calculated, and a weld bead occurs only around the groove. See **Figure 20-14**.

Figure 20-12.
Select the appropriate fill direction to create the desired groove weld.
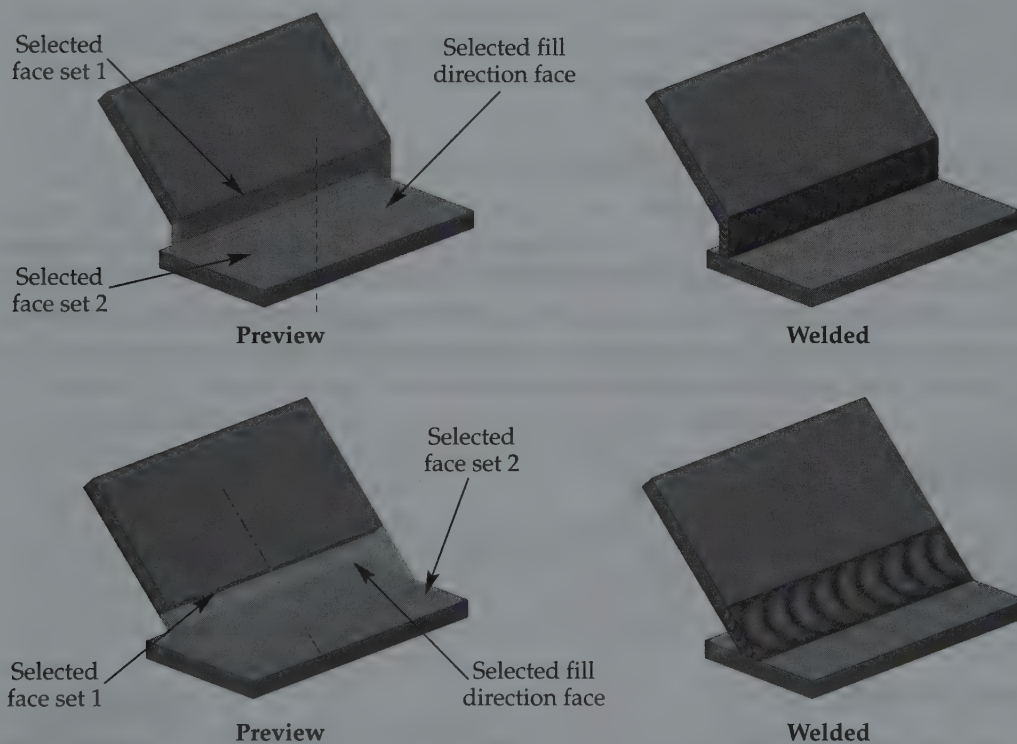
Figure 20-13.
An example of a case in which using the **Radial Fill** option is suitable.
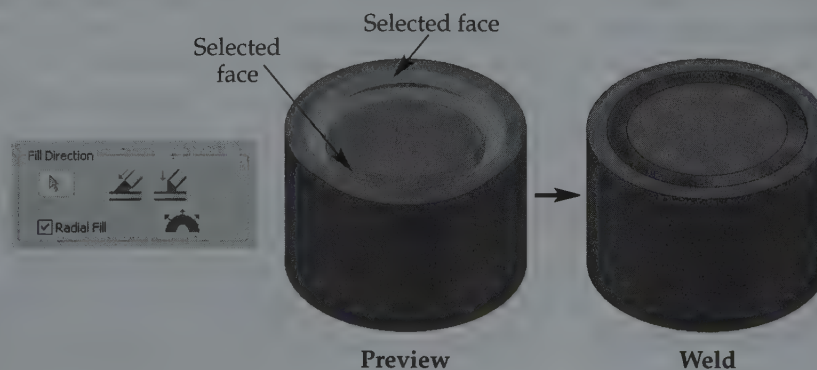
Figure 20-14.
These groove welds illustrate the effects of using the **Ignore Internal Loops** option.
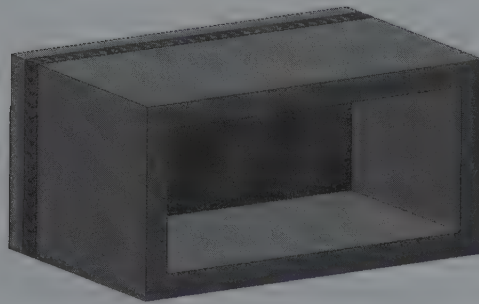
Without Ignore Internal Loops Option   With **Ignore** Internal Loops Option
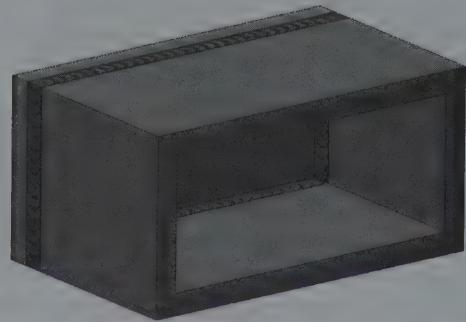
---

PROFESSIONAL TIP

Help locate geometry to constrain using the **Select Other** tool and wheel mouse selection techniques. Do not rely on picking both **Full Face Weld** check boxes to define the fill direction. Picking both **Full Face Weld** check boxes can result in an error that you can solve by specifying a fill direction.

Adding a Welding Symbol and Finalizing the Weld

Pick the **Create Welding Symbol** check box to create a welding symbol for reference in the model and to extract to a weldment drawing. The **Create Welding Symbol** area functions the same as when adding fillet welds, except that no fillet weld linking is possible. Pick the appropriate weld symbol button and specify values using the welding symbol interface. Pick the **Apply** button to create the weld and remain in the tool, or pick the **OK** button to create the weld and exit.
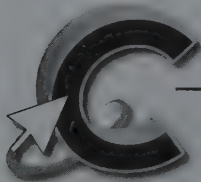
Exercise 20-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-5.

Cosmetic Welds

Access the **Cosmetic Weld** tool to create *cosmetic welds* using the **Cosmetic Weld** dialog box. See **Figure 20-15**. You can apply cosmetic welds to any joint to indicate the location and specifications of most welding operations. Use the **Select Edge(s)** button to select intersections of components at which to weld. When selecting multiple continuous tangent edges, pick the **Chain** radio button so you do not have to select edges individually. When selecting a closed loop, pick the **Loop** radio button so you do not have to select edges individually.

Ribbon

Weld
  > Weld

Cosmetic

NOTE

To deselect faces, with the **Select Face(s) 1** or **Select Face(s) 2** still active, hold down [Shift] or [Ctrl] and pick the faces to remove from the selection.

Figure 20-15.
Using the **Cosmetic Weld** dialog box to create a cosmetic weld.

Selected edges
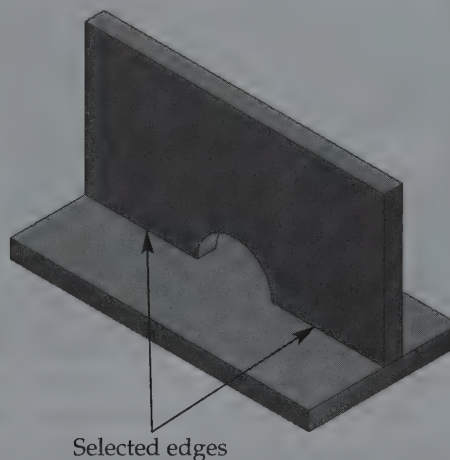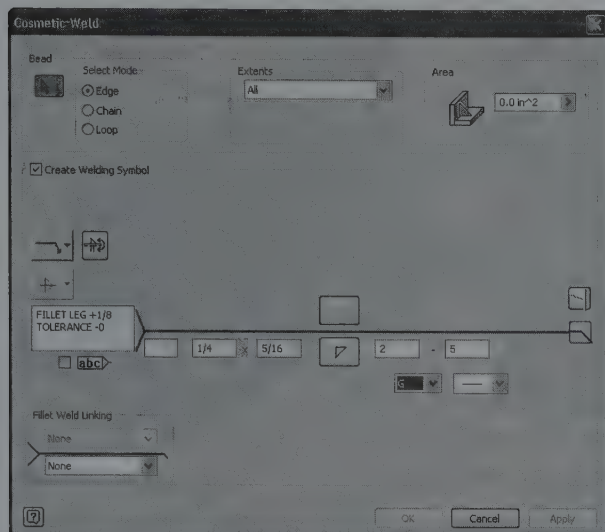
The **Extents** area allows you to define the extents of the weld. Pick the **All** option to weld along the entire joint. Pick the **From-To** option to weld between two parallel faces or planes selected using the **From** and **To** buttons. The **Area** option allows you to define the cross-sectional area of the weld bead, even though a weld bead does not appear in the model. The default value is 0 to the second power, but you can enter any acceptable number or equation.

Adding a Welding Symbol and Finalizing the Weld

Pick the **Create Welding Symbol** check box to create a welding symbol for reference in the model and to extract to a weldment drawing. Welding symbols are especially important to cosmetic welds because cosmetic welds do not contain specific model parameters like welds created using the **Fillet Weld** and **Groove Weld** tools. If you do not add a welding symbol, a cosmetic weld is just a representation that implies that some type of weld occurs at a joint.

The **Create Welding Symbol** area functions the same as when adding fillet welds, except that no fillet weld linking is possible. Pick the appropriate weld symbol button and specify values using the welding symbol interface. Pick the **Apply** button to create the weld and remain in the tool, or pick the **OK** button to create the weld and exit.

Exercise 20-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-6.

Welding Symbols

Welding symbols are available in a model for reference and are extractable to a weldment drawing. For most applications, you apply welding symbols when using the **Fillet Weld**, **Groove Weld**, and **Cosmetic Weld** tools symbols by picking the **Create Welding Symbol** check box and using the options in the **Create Welding Symbol** area. However, you can use the **Welding Symbol** tool to add a welding symbol to the model if you have not already established a welding symbol while creating a weld bead.

The **Welding Symbol** dialog box appears when you access the **Welding Symbol** tool. See **Figure 20-16**. Use the **Select Bead** button to pick a weld bead to apply a welding

Ribbon
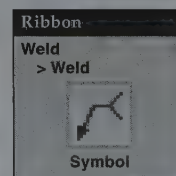
Weld
> Weld

Symbol

Figure 20-16.
An example of adding a welding symbol to a fillet weld using the **Welding Symbol** dialog box.

Selected bead
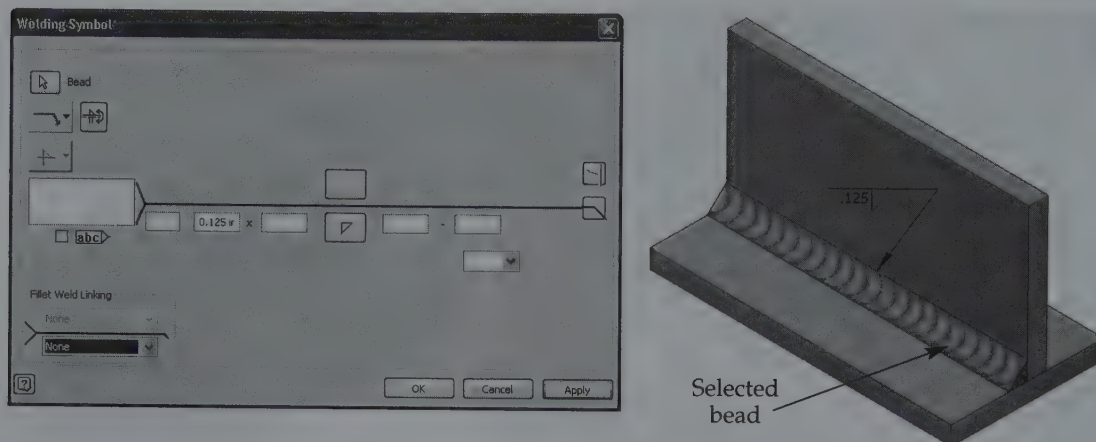
symbol. Then define the welding symbol using the rest of the **Welding Symbol** dialog box options. The remaining options in the **Welding Symbol** dialog box behave the same as those in the **Create Welding Symbol** area of the welding tools. Pick the appropriate weld symbol button and specify values using the welding symbol interface. Pick the **Apply** button to create the weld and remain in the tool, or pick the **OK** button to create the weld and exit.

End Fills

Access the **End Fill** tool to add an *end fill* for model representation purposes and to extract to a weldment drawing. See **Figure 20-17**. Then pick the end of the weld beads to modify. By default, when you create a fillet weld, an end fill is applied automatically to the weld bead. As a result, the end fill is removed when you pick the end of a fillet

Ribbon
Weld
> **Weld**
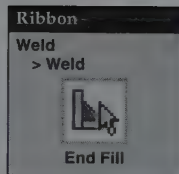
End Fill

end fill: End of a weld bead identification.

Figure 20-17.
Compare the display of the weld bead end without fill to the weld bead with fill.

Not filled

Filled
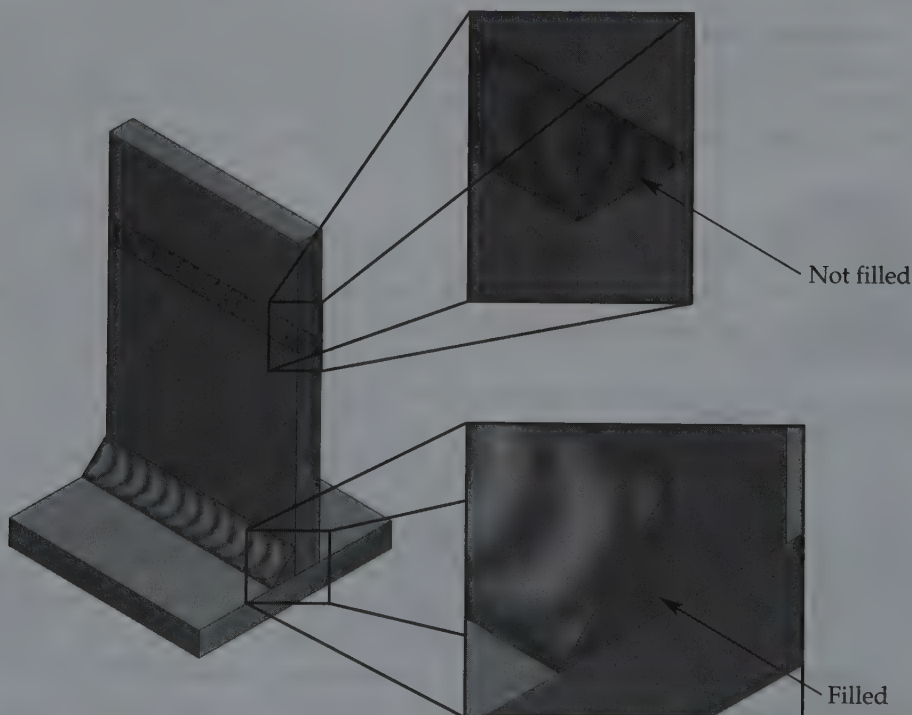
weld while using the **End Fill** tool. Groove welds do not automatically generate an end fill, so picking the end of a groove weld adds an end fill.

Managing Weld Beads

Modify weldments and weldment components using many of the same techniques as editing parts, subassemblies, and assemblies. Adjust preparations, welds, and machined features, described later in this chapter, using options in the browser. Weld beads and welding symbols, in particular, may require edit and include some specific management options.

The **Bead** folder in the **Welds** node of the browser stores weld beads. Drag **End of Features** in the **Bead** folder as needed to suppress weld beads. Welding symbols are listed separately in the browser and consume a copy of the model weld bead, although the weld bead is also displayed in the **Beads** folder. Right-click on the **Welds** folder or a weld bead in the browser or graphics window and deselect **Visibility** to turn off the display of weld bead geometry. Right-click on the **Welds** folder or a weld bead in the browser or graphics window and deselect **Symbol Visibility** to turn off the display of welding symbols, but not weld bead geometry. This is a common requirement when you are working with large or complex weldments that have many welding symbols.

Right-click on the **Welds** folder or a weld bead in the browser or graphics window and select **iProperties** to access the **iProperties** dialog box associated with weld beads. You have already learned the importance of using the **iProperties** dialog box to specify the correct weld bead material from the **Material** drop-down list in the **Physical** tab. The **Weld Bead** tab includes additional options that are unique to weldments. The **Visible** check box provides another means of toggling the display of weld bead geometry and welding symbols. Deselect the **Enabled** check box to disable selection and editing of weld beads. The **Weld Bead Color Style** drop-down list controls the color and display characteristics of weld beads. The **End Fill Color Style** controls the color of weld bead end fills. To display the .bmp image of weld bead geometry and use the corresponding weld filler material, you must choose the appropriate welded material from the **Physical** tab and select the **As Material** color style option.

NOTE

Turning off weld bead or welding symbol visibility does not actually remove welds or welding symbols from the model.

Machining

Weldment machining is required for any post-weld operation, such as adding a joint, groove, hole, or other feature. The type, location, and size of machined features are specific to the welding application and design. **Figure 20-18** shows examples of weldment machining operations. Although you can machine components in the assembly file, machining tools in the weldment work environment mimics the process of post-weld processes.

weldment machining: Any task performed after you weld assembly components, such as drilling a hole through components, preparations, and welds.

The **Machining** node in the browser stores machined features. Activate the machining mode from the ribbon, double-click on the **Machining** node in the browser, or right-click on the node and select **Edit**. The machining work environment provides common feature, work feature, feature pattern, and parameter tools appropriate for post-weld operations. These tools behave the same as those in the standard part model environment, except that you can only remove material when machining.

Ribbon
Weld
 > Process

Machining

Figure 20-18.
Examples of various weldment machined features.

Weldment before Additional Machining

Weldment after Additional Machining
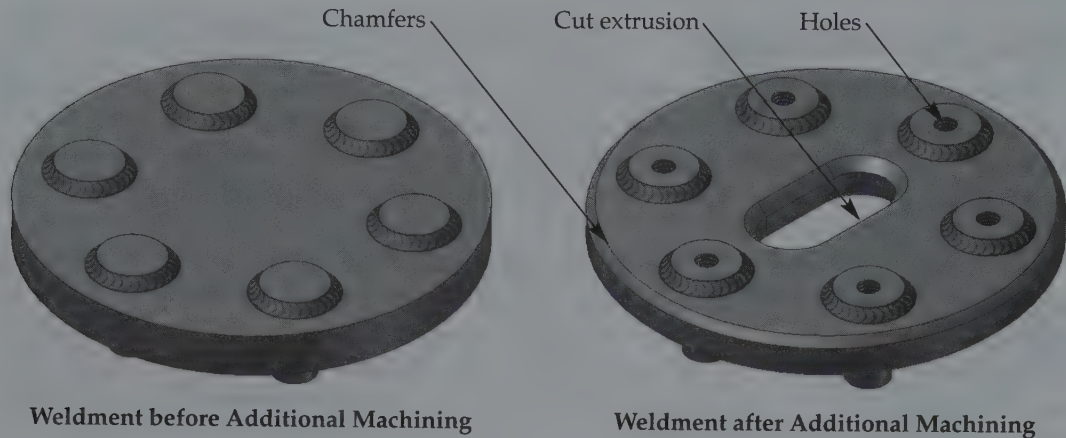
Use the **Extrude**, **Revolve**, and **Sweep** tools to cut a sketch profile, and use the **Hole** tool to add holes. Use the **Fillet** tool to cut rounds and the **Chamfer** tool to cut chamfers. Use the **Move Face** tool to move faces, and pattern features such as holes and slots using the **Mirror**, **Rectangular Pattern**, and **Circular Pattern** tools. When you are finished machining, exit the machining mode by double-clicking on the weldment icon in the browser or by activating another weldment process. You can also right-click on the **Machining** node and select **Finish Edit**, or use the **Return** tool. Re-enter the machining mode as needed throughout the weldment design process.

Exercise 20-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 20-7.

weld bead report: A list of all the welds added to the weldment, including data on the Inventor tool used to create the weld bead and physical properties, such as the length of each weld, volume, and mass.

Weld Bead Reports

Access the **Weld Bead Report** tool to export weld bead information as an .xls spreadsheet file to create a *weld bead report*. See **Figure 20-19**. Use a weld bead report to determine the volume and mass of weld beads and help calculate welding material usage and weld times.

The **Weld Bead Report** dialog box appears when you access the **Weld Bead Report** tool. Pick the **Include All Subassemblies** check box to include subassembly weld beads in the report. Then pick the **Next** button to access the **Report Location** dialog box, which allows you to export, or save, the weld bead report as a Microsoft Excel spreadsheet file.

Figure 20-19.
An example of a weld bead report.

	A	B	C	D	E	F	G	H	I	J	K
1	Document	ID	Type	Length	UoM	Mass	UoM	Area	UoM	Volume	UoM
2	C:\Motor Mount.iam										
3		Fillet Weld 2	Fillet	5	in	0.026	lbmass	0.018	in^2	0.092	in^3
4		Cosmetic Weld 2	Cosmetic	7	in	N/A		N/A		N/A	
5		Groove Weld 1	Groove	N/A		0.035	lbmass	N/A		0.125	in^3
6		Groove Weld 2	Groove	N/A		0.035	lbmass	N/A		0.125	in^3
7											

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. Define *welding*.
2. What is a weldment?
3. Briefly describe material preparation for welding.
4. What is a root opening?
5. What is the weld placement, or welding process?
6. Briefly describe a fillet weld.
7. In a weld, what is a leg?
8. Define *throat*.
9. Describe a contour.
10. What are toes?
11. Define *pitch*.
12. Explain the meaning of *arrow side* and *other side*.
13. Explain the appearance and basic function of the reference line.
14. What is a weld symbol?
15. What are groove welds?
16. Briefly describe fill direction.
17. Describe cosmetic welds.
18. What is an end fill?
19. Explain the basic function of weldment machining.
20. Describe the contents of a weld bead report.

Problems

1–3 Instructions:

- *Open the specified assembly file and save the file as the given name. Convert the assembly to a weldment. Select the appropriate weld bead material during the conversion.*
- *Follow the specific instructions for each problem to create the weldment.*

1. **File:** P18-3.ipt
   Save as: P20-1.ipt
   Title: HOUSING ASSEMBLY
   Weld bead material: Welded Steel Mild
   Project: HOUSING
   Save as: P18-3.iam
   Specific Instructions: Place two .12″ × .12″ chamfer preparations to create the bevel-grooves shown. Place one chamfer on the edge of each side of the housing, not the housing cover. Add the groove welds with welding symbols shown to each side of the housing.
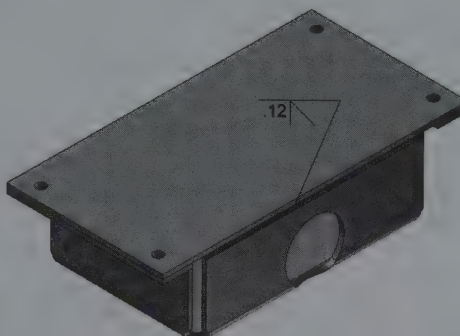
2. **File:** EX19-1.ipt
   Save as: P20-2.ipt
   Title: STUDDED HUB
   Weld bead material: Aluminum-6061
   Specific Instructions: Add the fillet welds with the welding symbols shown. Notice that the welds are on both sides of the feature. Select the top and bottom face of the cylindrical plate as the first selection (two faces total), and then pick the large and small cylindrical stud faces as the second selection (12 faces total).
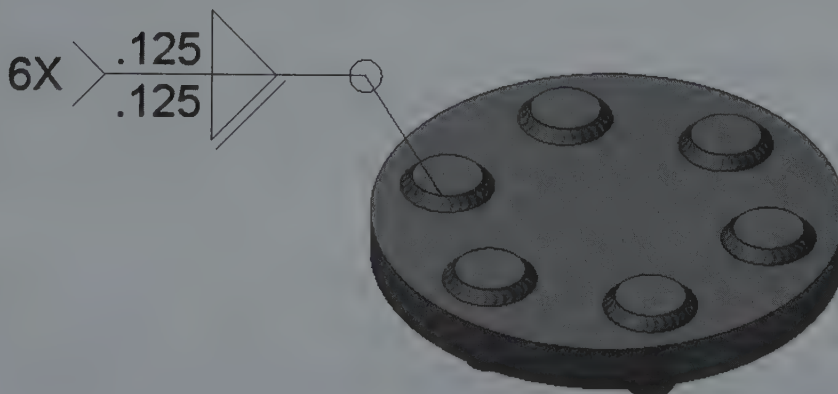
3. **File:** P20-3ASSY.iam
   (This file and its component .ipt files are provided on the Student Web site at www.g-wlearning.com/CAD.)
   Save as: P20-3.ipt
   Title: ALIGNMENT BAR
   Weld bead material: Welded Stainless Steel, 440C
   Specific Instructions: Place the four .0625″ × .0625″ chamfer preparations to create the V-grooves shown. Add the cosmetic and fillet welds with welding symbols shown.
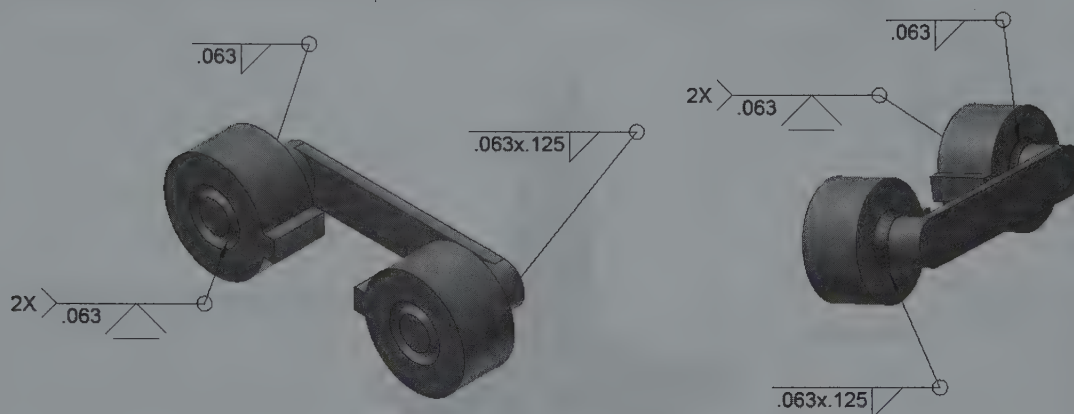
Problem 4 Instructions:

- *Prepare new components for insert or develop components in place to create the following weldment.*
- *Use appropriate sketching techniques and fully constrain sketch geometry.*
- *Add all necessary assembly constraints.*
- *Use your own judgment and approximate dimensions when necessary.*
- *Add as much information as possible to the **iProperties** dialog box for component and assembly files. Assign the specified material and color to parts.*
- *Follow the specific instructions for each problem to create the assemblies.*

Problems – Chapter 20

4. **Title:** STAND
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-109
   Project: STAND
   Material: Steel, Mild
   Color: As Material
   Save as: P20-4.iam
   Specific Instructions: Design a stand weldment similar to the example shown. Add preparations, welds, and machined features as required, using your own specifications.
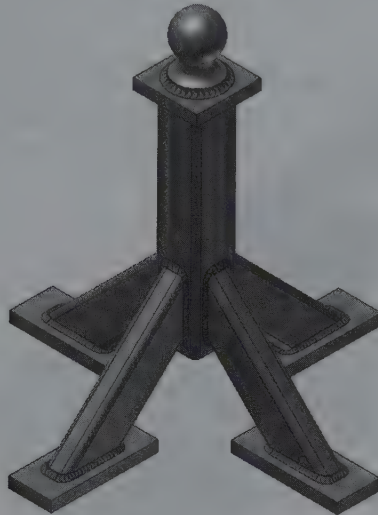
Presentations

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create and explode presentation views.
✓ Edit tweaks and trails.
✓ Use the **Precise View Rotation** tool.
✓ Animate presentations.

Use a *presentation* file to record design intent, help with assembly visualization, and show how components fit together and interact with other components. You can also use a presentation to create an *exploded assembly* drawing, also known as an *illustrated parts breakdown*. Common presentation applications include illustrating instruction manuals, identifying maintenance and assembly procedures, and creating assembly animations. **Figure 21-1** shows an example of a presentation model.

presentation: An exploded and animated assembly or weldment model.

exploded assembly (illustrated parts breakdown): An assembly with components moved away from other components in an unassembled or ungrouped view, but usually maintaining alignment.

Figure 21-1.
An example of a presentation model created by referencing and then exploding, or tweaking, an assembly model.
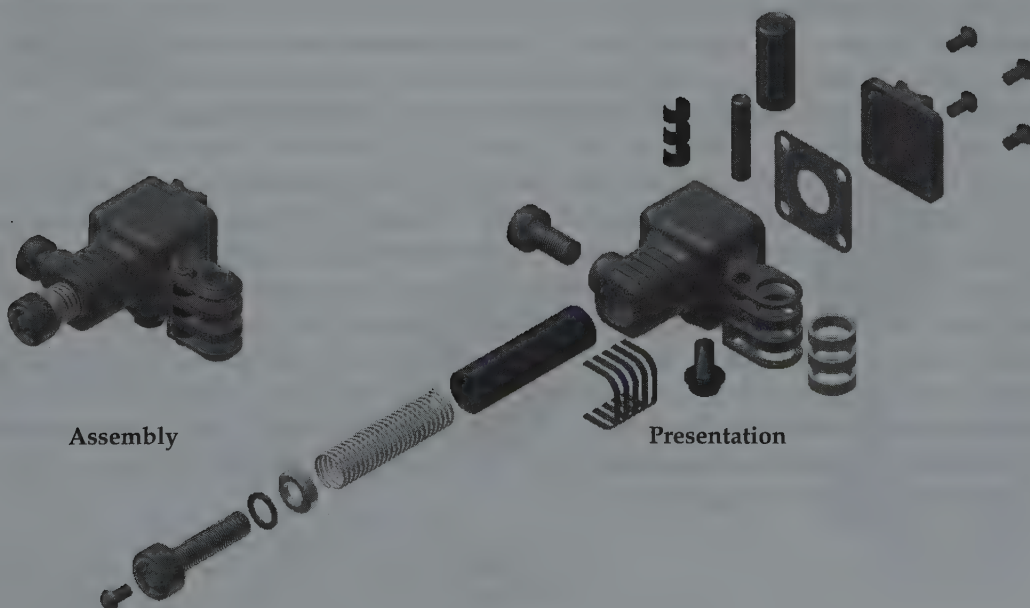
Assembly

Presentation

Presentation files carry the extension .ipn. The presentation work environment provides tools and options that allow you to create presentation views by referencing, exploding, rotating, and animating assembly models. Each presentation file can include multiple presentation views to display the model in different configurations. For example, you can use different views to display various assembly or disassembly options. Each presentation view references the same assembly file. You must use another presentation file to create a presentation model of a different assembly.

> **NOTE**
>
> Presentation files only provide a means of displaying assemblies. They do not provide the ability to modify component parameters. You must edit an assembly or the assembly components to make changes to the model.

Exercise 21-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-1.

Exercise 21-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-2.

Initial Presentation View

Ribbon
Presentation
> Create

Create View

Shortcut
Create View...
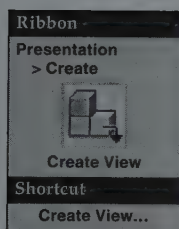
Access the **Create View** tool to place the initial presentation view using the **Select Assembly** dialog box. See **Figure 21-2A**. Pick the **Browse** button to locate and select an assembly file. Then, if necessary, pick the **Options...** button to select a design view, positional, and level of detail representation using the **File Open Options** dialog box. See **Figure 21-2B**.

Select the **Public** radio button to choose a design view representation stored in the assembly file from the drop-down list. Pick the **Private** radio button to use a design view representation in a separate .idv file. The **Browse** button allows you to locate the .idv file and then select the design view representation from the drop-down list. Check **Associative** after you select a design view representation to link the design view representation with the presentation, so that changes made to the design view representation automatically apply to the presentation. Select a positional representation from the **Positional Representation** drop-down list, and pick a level-of-detail representation using the **Level of Detail Representation** drop-down list. Pick the **OK** button to return to the **Select Assembly** dialog box.

Figure 21-2.
A—The **Select Assembly** dialog box. B—The **File Open Options** dialog box.

Pick to open the **File Open Options** dialog box

Pick to locate an assembly file
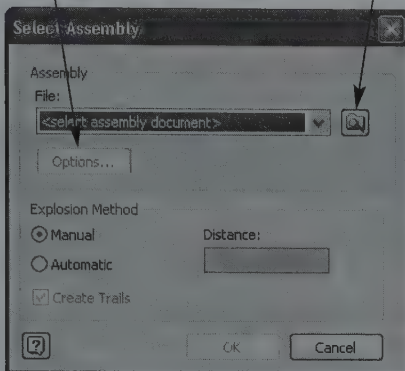
A
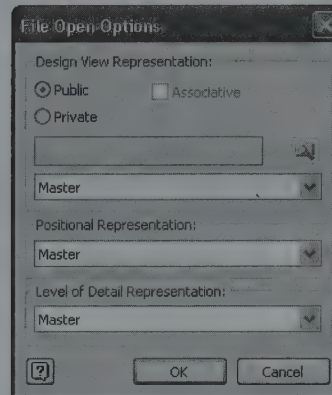
B

Pick the **Manual** radio button to explode assembly components in the presentation view using the **Tweak Components** tool, described later in this chapter. An alternative is to select the **Automatic** radio button to allow Inventor to explode, or *tweak*, components during view placement. Specify the *explosion distance*, or *tweak distance*, using the **Distance** text box. For example, the cylinder shown in **Figure 21-3** has a 1″ tweak distance. Check **Create Trails** to include *trails* with the automatic explosion. Pick the **OK** button to create the presentation view.

> **PROFESSIONAL TIP**
>
> Manual explosion is the most common explosion practice. Automatic explosion tweaks all components the same distance, and according to assembly constraints. The result is often not correct, especially for large or complex assemblies.

tweak: Inventor's terminology for the process of moving, or exploding, an assembled component into an exploded assembly.

explosion distance (tweak distance): The distance the component moves from the assembled view during explosion.

trails: Lines that indicate the bond between components and the assembly by displaying the distance and location of the explosion.
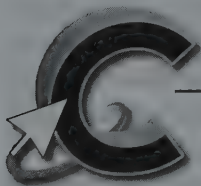
Exercise 21-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-3.

Figure 21-3.
An example of an automatically exploded presentation with a trail to indicate the distance and direction of travel.

Trail

Tweak distance

Assembled Presentation

Exploded Presentation

Exercise 21-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-4.

Exploding

If you choose the **Manual** radio button when you create the view, the model appears in the assembled form, and you are responsible for explosion. To explode an unexploded view automatically, expand the presentation view (Explosion1 by default) in the browser, right-click on the assembly file, and select **Auto Explode**. The **Auto Explode** dialog box appears, as shown in **Figure 21-4**.

Specify the tweak distance using the **Distance** text box and check **Create Trails** to include trails. Then pick the **Preview** button to display the intended explosion. If the preview looks acceptable, pick the **OK** button to apply the settings and explode the assembly. An automatic explosion separates and aligns all the assembly components at one time, using a specified distance between the components. Tweaks are specified automatically when you automatically explode an assembly in the presentation file. When you manually explode an assembly, you tweak individual components from other components.

Exercise 21-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-5.

Adding Tweaks

Though automatic explosion is sometimes quick and effective, manual explosion is often more appropriate and provides greater control and accuracy. In general, apply tweaks in the reverse order of assembly processes. For example, the first tweak should be the last process required to create the assembly. Access the **Tweak Components** tool to tweak components using the **Tweak Component** dialog box. See **Figure 21-5**.

The **Direction** button is initially active, allowing you to pick an edge, face, or feature to place a triad that you can use to identify the *tweak direction*. See **Figure 21-6**. The highlighted triad axis indicates the axis along which the component moves. Pick a different axis or use the **X**, **Y**, or **Z** button in the **Transformations** area to change the tweak direction. Use the **Direction** button as needed to establish a more appropriate triad.

Ribbon
Presentation
> Create

Tweak Components
Shortcut
Tweak
Components...

tweak direction:
The direction in which exploded components move.

Figure 21-4.
The **Auto Explode** dialog box.

Auto Explode

Distance
1.000   Preview

☑ Create Trails

②   OK   Cancel

Figure 21-5.
The **Tweak**
Component dialog
box.
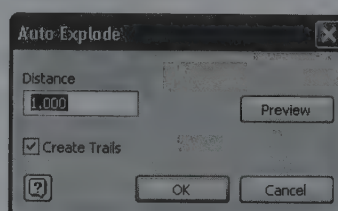
Figure 21-5. The **Tweak Component** dialog box.
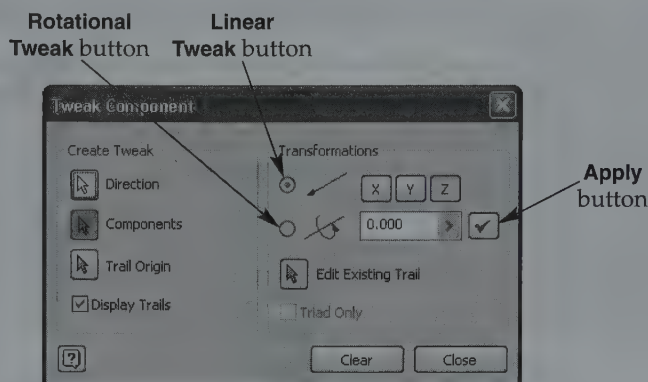
Figure 21-6.
Select an edge, face, or feature, such as the cylindrical feature shown here, to place the direction triad. The highlighted axis indicates the direction of travel.

Selecting the
Tweak Direction

The Selected
Direction
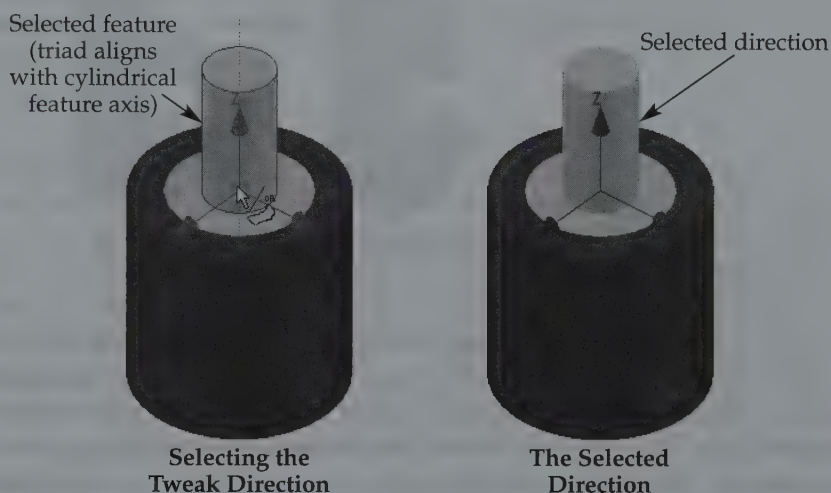
The **Components** button becomes active, allowing you to select the component(s) to tweak. A component is preselected if you accessed the **Tweak Components** tool by right-clicking on a component in the browser or graphics window. Select all the components to tweak during the same operation. Deselect a component by holding down [Ctrl] and picking the component to remove from the tweak.

Figure 21-7 shows an example of selecting multiple components for the first tweak and a single component for the second tweak. In general, the first tweak includes the most components, and additional tweaks include fewer and fewer components until there are no more components to tweak. Pick the **Trail Origin** button and select a point on the assembly to identify the origin, or beginning, of the trail from which tweaked components originate. If you do not specify a trail origin, the trail begins at the selected component's center of mass. Select the **Display Trail** check box to show a trail.

The **Transformations** area controls all other tweak parameters. Pick the **Linear Tweak** radio button to tweak the selected components along the specified axis. See **Figure 21-8**. If you activate the **Components** button, you can drag the selected components to adjust the tweak distance. To apply a precise distance, specify a value in the **Tweak Distance** text box. Pick the **Apply** button or press [Enter] to create the linear tweak.

Pick the **Rotational Tweak** radio button to rotate the selected components around the specified axis. You often apply a rotational tweak with a linear tweak to make components revolve around and travel along the axis, such as when a nut threads onto a bolt. See **Figure 21-8**. If you activate the **Components** button, you can drag the selected components to adjust the tweak angle. To apply a precise angle, specify a value in the **Tweak Angle** text box. Pick the **Apply** button or press [Enter] to create the rotational tweak.

Figure 21-7.
This example shows tweaking a C-clamp screw and pin together, and then applying another tweak to explode the pin from the screw.

Assembled   First Tweak   Second Tweak
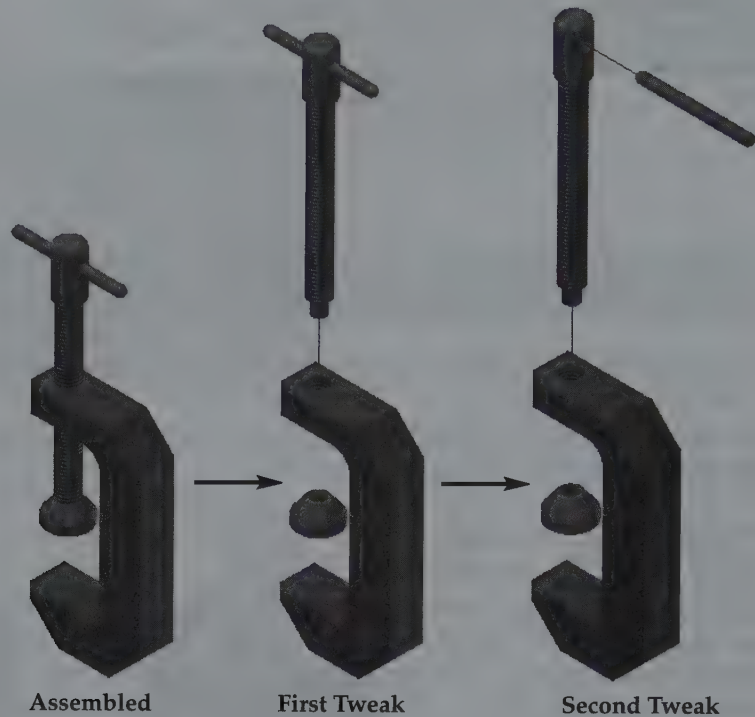
The **Triad Only** check box is available when you pick the **Rotational Tweak** radio button as an option for rotating the triad without rotating components. This allows you to adjust the available axes when the selected direction does not provide an appropriate axis. Drag the triad to adjust the angle, or specify a precise angle in the **Tweak Angle** text box. Pick the **Apply** button or press [Enter] to rotate the triad.

When you apply a tweak or triad rotation, the **Tweak Component** dialog box remains open with the same selected direction, components, and trail origin. This is a convenient way to continue applying tweaks to the same components. You can adjust tweak parameters, such as selecting a different axis, and with the **Components** button active, hold down [Ctrl] to deselect components. Then continue applying tweaks as needed. Pick the **Clear** button to reset the **Tweak Component** dialog box to start fresh when applying additional tweaks. When you are finished, pick the **Close** button to exit.

Figure 21-8.
Linear tweaks move items such as the washers and nut along an axis. Rotational tweaks show rotation, such as the nut rotating as it threads on the bolt.

Linear tweaks

Assembled

Rotational tweak

Exploded
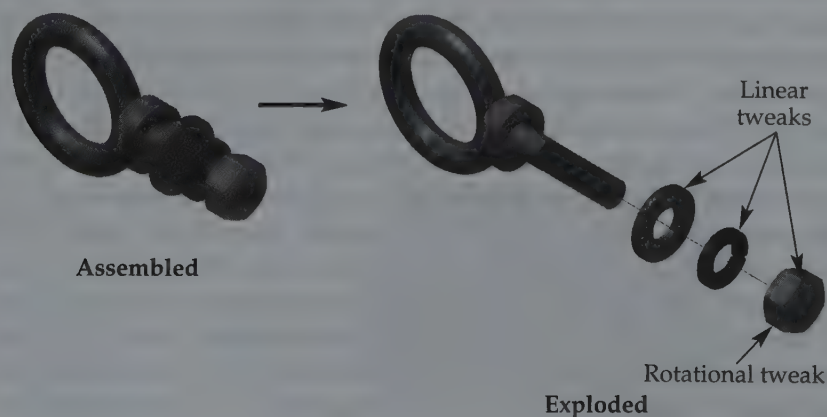
The **Assembly View** browser filter option is set as default and displays the assembly hierarchy. The presentation view is listed first, followed by the assembly file, and then each assembly component. Tweaks are listed under specific components. Select the **Tweak View** browser filter option to display a tweak hierarchy, which clearly identifies the explosion characteristics of the presentation. The tweak specifications used to explode the assembly in each presentation view appear first, followed by the assembly and assembly components.

Exercise 21-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-6.

Editing Tweaks and Trails

Editing tweaks and trails is a common task. The location of tweak and trail editing options depends on the browser filter setting. The **Sequence View** filter allows you to modify tweak sequence parameters that are unavailable when the **Assembly View** filter is active. However, you typically use sequence parameters for animation purposes, as described later in this chapter. The following information focuses on general tweak and trail modification tools.

Right-click on a tweaked component in the browser or graphics window to access a menu of options for managing tweaks associated with the component. Use the **Delete Tweaks** cascading submenu to delete individual tweaks using the **Last** option, or delete all tweaks using the **All** option. See **Figure 21-9**. Pick **Add Trail** to select points on

Figure 21-9.
Right-click on a component and use an option from the **Delete Tweaks** cascading submenu to remove tweaks.

Presentation with Two Tweaks

Right-Click and Select Delete Tweaks

Last Tweak Deleted

All Tweaks Deleted
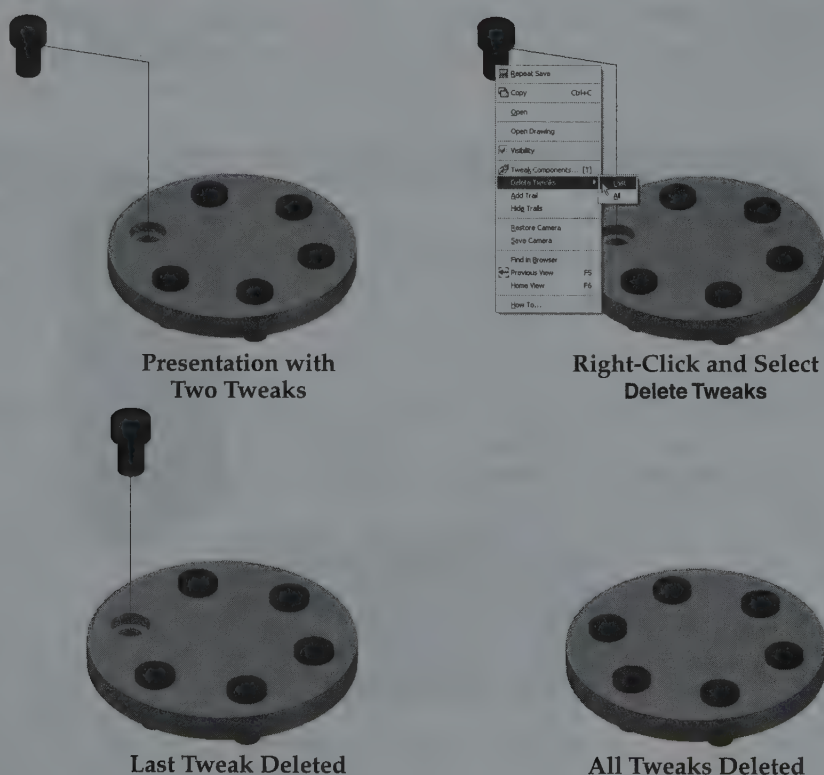
the component to add tweak trails. Right-click and choose **Reselect** to pick different points. Right-click and select **Done** to generate the new trails. See **Figure 21-10**. Pick **Hide Trails** to "turn off" the visibility of tweak trails.

You can also right-click on a tweak trail in the graphics window to access a menu of options for adjusting tweaks and trails. The **Hide Trails** option is available as previously described. Pick **Delete** to remove the tweak, not just the trail. Use the **Visibility** option to turn off display of visibility of the selected trail. The **Edit...** option reopens the **Tweak Component** dialog box for editing the selected tweak.

> **NOTE**
>
> You can also reopen the **Tweak Component** dialog box to edit a tweak by double-clicking on a component or component trail. The **Delete** and **Visibility** options are also available when you right-click on a tweak in the browser.

You can edit a tweak using the **Tweak Component** dialog box. However, easier options are available for adjusting the tweak distance or rotational angle. One option is to pick the trail of the tweak and then drag the green dot to the desired location. See **Figure 21-11A**. Another option is to pick a tweak in the browser and adjust the distance or angle using the edit box shown in **Figure 21-11B**.

Exercise 21-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-7.

Figure 21-10.
An example of adding trails to a tweak for better clarity.

Selected points

Additional trails

Original trail
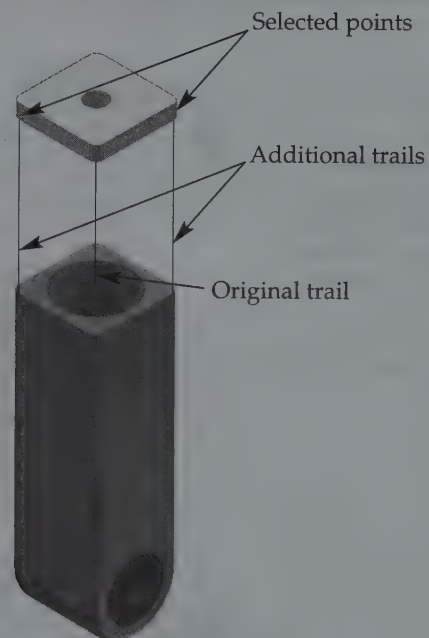
Figure 21-11.
A—Dragging a component to change the tweak distance. B—Using the text box to modify a tweak amount. This figure also shows the filter options available for adjusting the display and organization of items in the browser.

Click and drag the green dot

Select **Assembly View**

Pick a tweak

Modify the tweak in this edit box

B

A
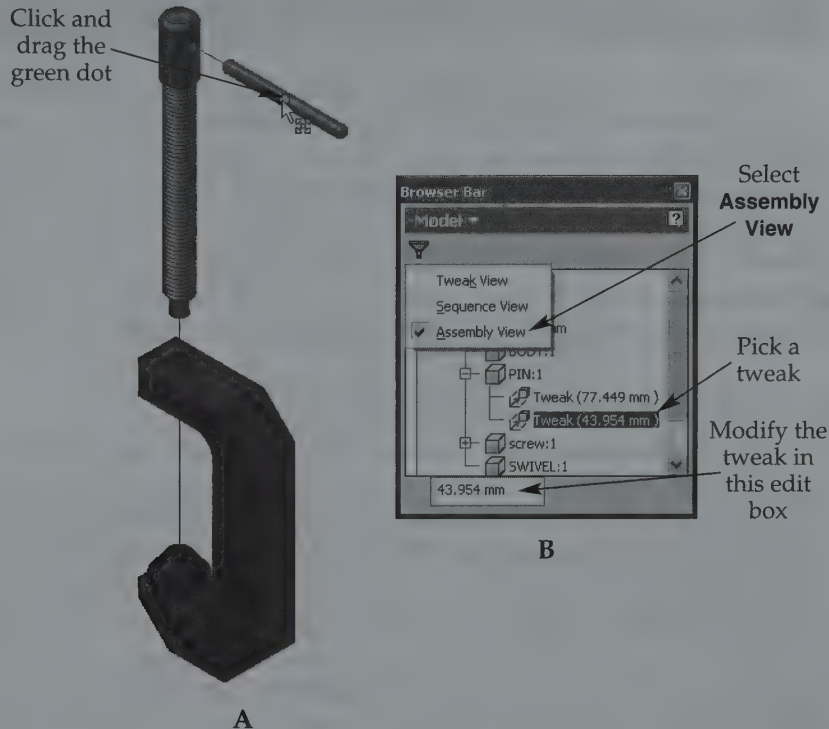
Precise View Rotation

The presentation environment contains view tools for modeling and for setting the *camera view*. It also includes the **Precise View Rotation** tool that allows you to orbit the view by an incremental value, such as 10° clockwise. The **Incremental View Rotate** dialog box appears when you access the **Precise View Rotation** tool. See **Figure 21-12**.

Specify the angle of rotation using the **Increment** text box. Then pick the appropriate rotate buttons to rotate the view. Each time you select a rotate button, the view rotates according to the specified direction and increment. Pick the **Reset** button to return to the original view. Pick the **OK** button to exit.

camera view: The position and zoom parameters of the presentation view; allows the view to change during animation.

Ribbon
Presentation > Create

Precise View Rotation

Shortcut
Precise View Rotation...
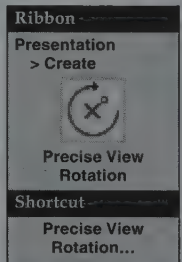
PROFESSIONAL TIP

After you define a specific presentation camera view using the **Precise View Rotation** tool, right-click and select **Save Camera** to save the current presentation view display.

Figure 21-12.
The **Incremental View Rotate** dialog box has buttons to control the direction of view rotation and an **Increment** text box to control the amount of view rotation.

Rotate Right

Roll Counter-clockwise

Roll Clockwise

Increment:

Reset   OK   Cancel

Rotate Down

Rotate Up

Rotate Left
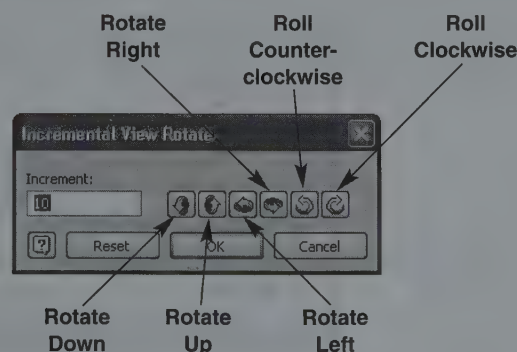
Animate a presentation view to show how components fit together or to display product assembly and disassembly characteristics. Common animation applications include maintenance and assembly manuals and general design review requirements. The **Animate** tool allows you to define, play, and record a presentation view animation. The options in the **Sequence View** browser provide another way to define and view an animation, but these options do not provide the advanced animation functions, including recording, that are available with the **Animate** tool. Often you will use both the **Sequence View** browser and the **Animate** tool to prepare an animation.

Browser Sequence Views

sequence: A group of tweaks that identify tweak relationships between components.

task: A group of sequences that define a certain assembly or disassembly relationship.

Pick the **Sequence View** browser filter option to help develop *sequence* and *task* specifications, especially for animating presentation views. Tasks describe how groups of components function in the animation, and sequences describe how each component functions in reference to other components during the animation. For example, **Figure 21-13** shows a presentation with two tasks. The first task includes three sequences. The first sequence aligns the washer with the bolt shaft, the second sequence aligns the lock washer with the bolt shaft, and the third sequence aligns the nut with the bolt shaft. The second task also contains three sequences, which assemble the washer, lock washer, and nut with the threaded eyebolt shaft as specified with tweaks.

NOTE

Although the presentation in **Figure 21-13** contains two tasks, you can place all sequences in one task. The sequences are not separate, as evidenced by the sequence numbers 1–6. You typically use multiple tasks for larger assembly presentations, such as when animating distinct components or subassemblies.

The key to effective animation is to manage tweaks, tasks, and sequences. To view the effects of a task or sequence, right-click on the task or sequence in the browser, and select **Edit...** to display the **Edit Task & Sequences** dialog box. See **Figure 21-14**. If necessary, select a different task to edit using the **Task** drop-down list and a different

Figure 21-13.
An example of a presentation with two tasks.
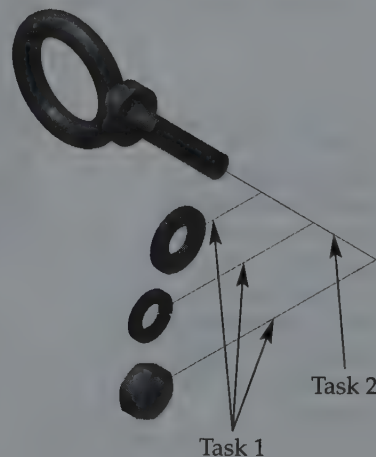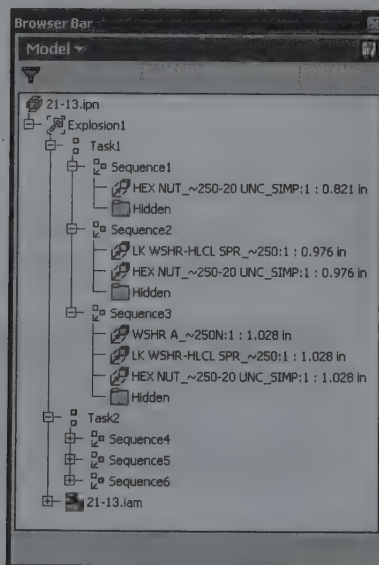
Figure 21-14.
The **Edit Task & Sequences** dialog box.

Task
Play Forward
button

Sequence
Play Forward
button
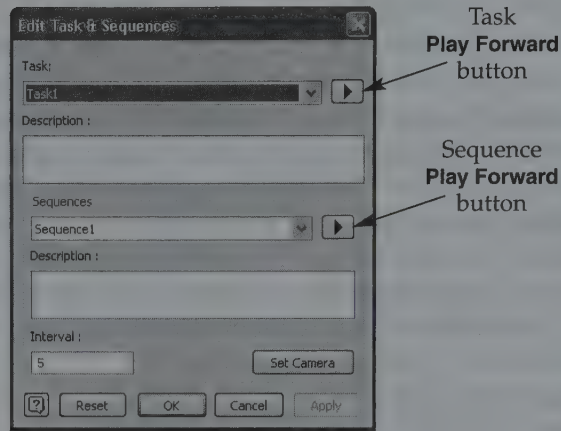
sequence to edit using the **Sequence** drop-down list. Enter a description of the animation task or sequence in the corresponding **Description** area. Descriptions typically clarify the process for future task and sequence editing reference. To observe the task or sequence, pick the task or sequence **Play Forward** button. Pick the **Reset** button to return to the exploded view.

You can also use the **Edit Task & Sequences** dialog box to specify the *sequence interval* and sequence camera view. The lower the value you specify in the **Interval** text box, the faster the playing speed. When you define a sequence camera view and then animate the presentation, the display automatically zooms in or out and rotates according to the specified camera view. To create a sequence camera view, rotate and zoom the presentation using viewing tools, including the **Precise View Rotation** tool. Then pick the **Set Camera** button to create the sequence camera view. Pick the **Apply** button to apply changes, and then continue working with the dialog box, or pick the **OK** button to exit.

To create a new task, right-click on the presentation view in the browser and select **Create Task**. Once the new task is present, you can add sequences by creating additional tweaks or by dragging existing sequences into the new task. To create a new sequence if a sequence contains more than one tweak, right-click on a tweak and select **New Sequence**. The new sequence becomes the first sequence of the first task and includes the specified tweak. As a result, the original sequence loses the tweak assigned to the new sequence.

Modifying sequence order is one of the most useful purposes of the **Sequence View** browser filter option, which allows you to redefine animation operations. To edit sequencing, drag the sequence above or below another sequence, depending on the requirement. For example, if a washer that should be assembled after a washer is Sequence1 and a lock washer that should be assembled before the lock washer is Sequence2, drag Sequence2 above Sequence1. In this example, Sequence2 becomes Sequence1. To group sequences so that the tweaks of two or more sequences occur at the same time, hold down [Ctrl] and select each of the sequences to group. Then right-click and select **Group Sequence**. A common application for grouping sequences is a component with linear and rotational tweaks that rotates and moves along the axis at the same time.

sequence interval:
The animation playing speed.

Exercise 21-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-8.

Animate Tool

Access the **Animate** tool to animate a presentation using the **Animation** dialog box. See **Figure 21-15**. Pick the **More** button to set the animation sequence using the options in the **Animation Sequence** area. Sequence numbers represent the order in which the components and corresponding tweaks animate. Pick a sequence to highlight the corresponding tweak trail in the graphics window. Components with the same sequence number indicate a task and animate together during the sequence. To separate sequences, select the sequences and pick the **Ungroup** button. To combine sequences, select the sequences and pick the **Group** button. Use the **Move Up** and **Move Down** buttons to change sequencing.

Once you set appropriate animation sequencing, you are ready to animate the presentation. Use the **Interval** text box to specify the sequence interval. The default *animation repetition* of 1 allows you to view the assembly or disassembly animation one time. Specify the number of repetitions using the **Repetitions** text box.

animation repetition: The number of times the animation goes through the animation process.

> **NOTE**
>
> After you modify animation parameters such as sequence settings, interval, or repetition, you must pick the **Apply** button before you can set the animation in motion.

Animating

The **Motion** area allows you to play and record the animation. Pick the **Forward By Tweak** button to move the animation forward by one tweak, and pick the **Reverse By Tweak** button to move the animation back by one tweak. Select the **Forward By Interval** button to advance the animation forward by one interval step, and select the **Reverse By Interval** button to move the animation back by one interval step.

Pick the **Play Forward** button to play the entire animation forward, and pick the **Play Reverse** button to play the animation in reverse. The **Auto Reverse** button allows you to play the animation forward, then after the forward cycle ends, automatically play the animation in reverse. You can move an animation in reverse only if you have advanced the animation forward. Use the **Pause** button to stop the animation.

Figure 21-15.
The **Animation** dialog box.
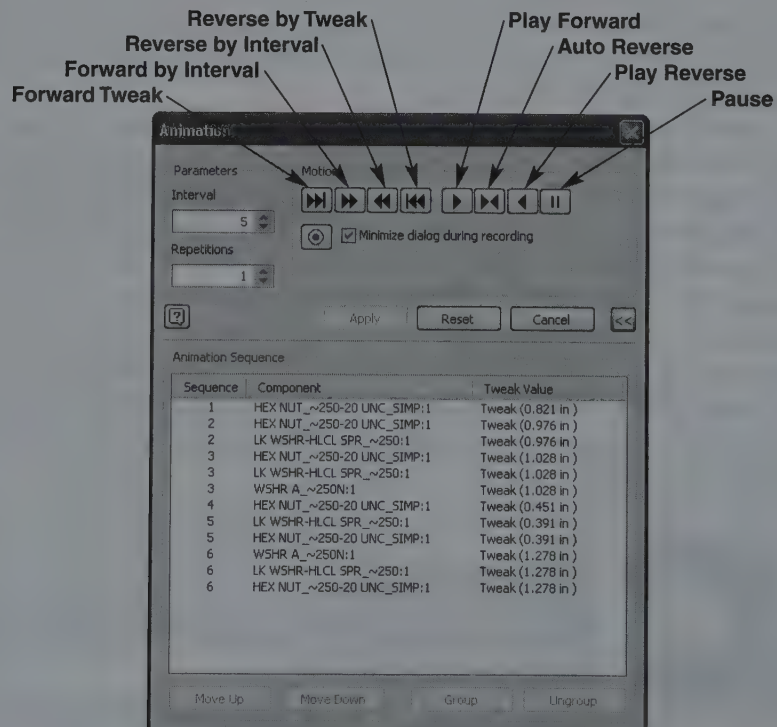
Recording and Finalizing

Pick the **Record** button to record the animation as an animation file. The **Save As** dialog box appears, allowing you to specify a location for the animation file. You can also choose the file format by selecting AVI or WMV from the **Save as type:** drop-down list. An additional dialog box appears, allowing you to specify animation file parameters. When you return to the **Animation** dialog box, pick the **Minimize dialog during recording** check box to minimize the **Animation** dialog box while you are recording the animation so that it does not appear in the animation.

Pick the **Reset** button to reset the animation and return the presentation view to its original display. When you are finished, pick the **Cancel** button to exit the **Animate** tool.

Exercise 21-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-9.

Adding Presentation Views

A presentation file can contain as many presentation views of the same assembly as needed. Typically, additional presentation views allow you to display different views of the assembly and create different types of animations. For example, you can show an assembly with unique component colors, or with some components suppressed. Another example is displaying an animation of an alternative assembly technique.

Access the **Create View** tool to place an additional presentation view using the **Select Assembly** dialog box. The **File** drop-down list and file **Browse** button are unavailable, because the assembly file used for the presentation was defined when you created the initial presentation view. You can pick the **Options...** button to select a design view, positional, and level of detail representation using the **File Open Options** dialog box. You also have the option of exploding the new view.

The active presentation view appears in the graphics window. Inactive views are shaded in the browser. To activate a different view, right-click on an inactivated view and select the **Activate** menu option, or double-click on the view name in the browser. As with other Inventor applications, you may find it helpful to rename presentation views in the browser.

PROFESSIONAL TIP

Predefined assembly representation views can be very useful for creating additional presentation views. Right-click on an item in the browser or the graphics window and select **Restore Camera** to return the presentation view display to the previously saved camera view. Pick **Save Camera** to set the display as the default view, which allows you to identify the desired position and zoomed display of the presentation view. You can then return to the saved view if you make modifications, such as orbiting or zooming.

Exercise 21-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 21-10.

Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or go to the Student Web site (www.g-wlearning.com/CAD) and complete the electronic chapter test.

1. What is a presentation?
2. Describe an exploded assembly.
3. Identify the type of file from which an initial presentation view must be developed, and explain the reason.
4. What does exploding an assembly in the presentation environment allow you to see?
5. Define *tweaking*.
6. Describe the explosion distance.
7. What are trails?
8. What is the tweak direction?
9. Explain the meaning of the term *camera view.*
10. In what ways can an animation be useful?
11. Describe the basic function of the **Animate** tool.
12. What is a task in an animation?
13. Define *sequence.*
14. Briefly explain the key to effective animation.
15. Why is it a good idea to enter a description of the animation task or sequence in the corresponding **Description** area?
16. What is a sequence interval?
17. What are sequence numbers?
18. Define *animation repetition.*
19. How many views can a presentation file contain?
20. What tool do you use to place an additional presentation view into a presentation file?

Problems

Instructions:

- *Begin a new presentation file for each problem and create an initial presentation view using the specified assembly.*
- *Explode and animate the initial presentation view.*
- *Create at least one more presentation view.*
- *Explode, manipulate, and animate the additional presentation views so they differ from the initial view.*
- *Add as much information as possible to the **iProperties** dialog box for component and assembly files. Assign the specified material and color to parts.*

Note: Level of difficulty may vary depending on number and complexity of additional presentation views.

1. **Title:** PUSH PIN
    Units: Inch
    Template: Presentation-IN.ipn
    Part Number: IAA-004
    Project: PUSH PIN
    Assembly File: P17-3.iam
    Save as: P21-1.ipn

    ▼ Basic

2. **Title:** BOTTLE ASSEMBLY
    Units: Inch or Metric
    Template: Presentation-IN.ipn or Presentation-mm.ipn
    Part Number: IAA-014
    Project: BOTTLE
    Assembly File: P17-4.iam
    Save as: P21-2.ipn

    ▼ Basic

3. **Title:** FORK
    Units: Metric
    Template: Presentation-mm.ipn
    Part Number: IAA-025
    Project: FORK
    Assembly File: P18-2.iam
    Save as: P21-3.ipn

    ▼ Basic

4. **Title:** HOUSING ASSEMBLY
    Units: Inch
    Template: Presentation-IN.ipn
    Part Number: IAA-039
    Project: HOUSING
    Assembly File: P18-3.iam
    Save as: P21-4.ipn

    ▼ Basic

5. **Title:** WATCH BAND CLASP
    Units: Inch or Metric
    Template: Presentation-IN.ipn or Presentation-mm.ipn
    Part Number: IAA-103
    Project: WRIST WATCH
    Assembly File: P17-6.iam
    Save as: P21-5.ipn

    ▼ Intermediate

6. **Title:** ROTATING HINGE ASSEMBLY
   Units: Inch or Metric
   Template: Presentation-IN.ipn or Presentation-mm.ipn
   Part Number: IAA-042
   Project: ROTATING HINGE ASSEMBLY
   Assembly File: P18-4.iam
   Save as: P21-6.ipn
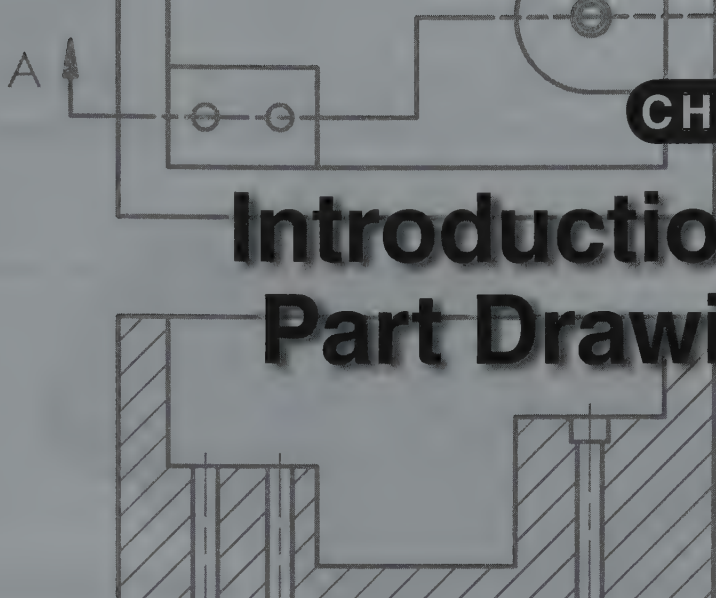
7. **Title:** SHOCK ABSORBER
   Units: Inch or Metric
   Template: Presentation-IN.ipn or Presentation-mm.ipn
   Part Number: IAA-106-001-01
   Project: UTILITY VEHICLE
   Assembly File: P19-4.iam
   Save as: P21-7.ipn

8. **Title:** SHOCK ABSORB SYSTEM
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-106-001
   Project: UTILITY VEHICLE
   Assembly File: P19-5.iam
   Save as: P21-8.ipn

9. **Title:** DISPENSER
   Units: Inch or Metric
   Template: Assembly-IN.iam or Assembly-mm.iam
   Part Number: IAA-107
   Project: DISPENSER
   Assembly File: P19-6.iam
   Save as: P21-9.ipn

Introduction to Part Drawings

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Describe methods of beginning and developing drawings.
✓ Apply correct ASME drafting standards.
✓ Create multiview drawings and auxiliary views.
✓ Extract section, broken-out section, and detail views.
✓ Draw conventional breaks and sketched views.
✓ Create and use sheet formats.

Inventor combines 3D solid modeling with 2D drawing capabilities. You can create part, assembly, and weldment drawings from existing models. **Figure 22-1** shows an example of a 2D part drawing created by referencing a 3D part model. When you edit a model, the corresponding drawing adjusts to reflect the new design. You can also edit a model by modifying parametric model dimensions inside a drawing. This chapter introduces drawing with Inventor and explores part drawing views. You will apply many of the same tools and options described in this chapter when developing assembly, weldment, and multiple sheet drawings.

Part Drawing Fundamentals

Inventor drawing tools and options allow you to create *detail drawings* by referencing part models. Common detail drawing applications include *monodetail* and *multidetail* drawings.

Inventor drawing files carry the extension .idw. AutoCAD drawing files carry the extension .dwg. You can prepare a drawing in Inventor using an .idw or .dwg file. The Inventor .idw and .dwg work environments are identical, but the .dwg format is for direct use in AutoCAD. This textbook focuses on using .idw files.

This textbook approaches drawing using the following steps: open a template with a blank *sheet*, specify sheet parameters, add a border and title block, place views, and add *annotations*. You can use a variation of this approach or apply a different method as appropriate. Inventor also offers the flexibility to make changes to existing drawings, such as selecting a different view scale or sheet size.

detail drawings: 2D drawings of individual parts containing all the views, dimensions, and specifications necessary to manufacture the part.

monodetail drawing: A drawing of a single part on one sheet.

multidetail drawing: A drawing of several parts on one sheet.

sheet: The base on which a drawing is created; represents the physical limits of the drawing area, or the paper size.

annotation: Letters, numbers, words, and notes used to describe information on a drawing.

Figure 22-1.
An example of a two-dimensional drawing created by referencing a three-dimensional model.

3D Solid Model

NOTES:

1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
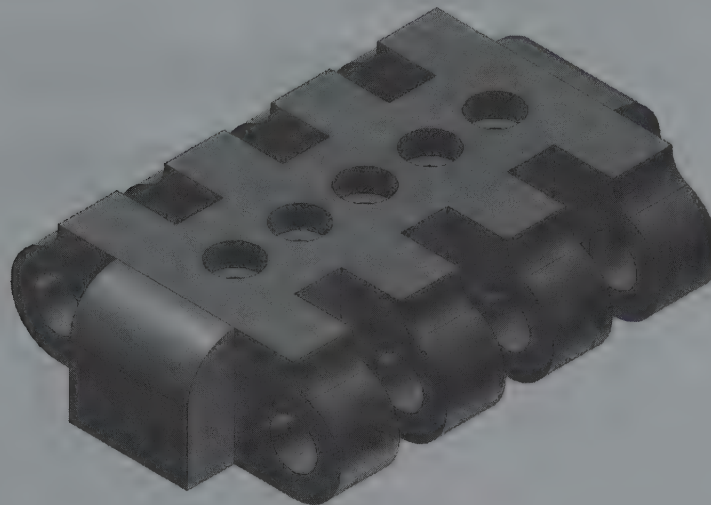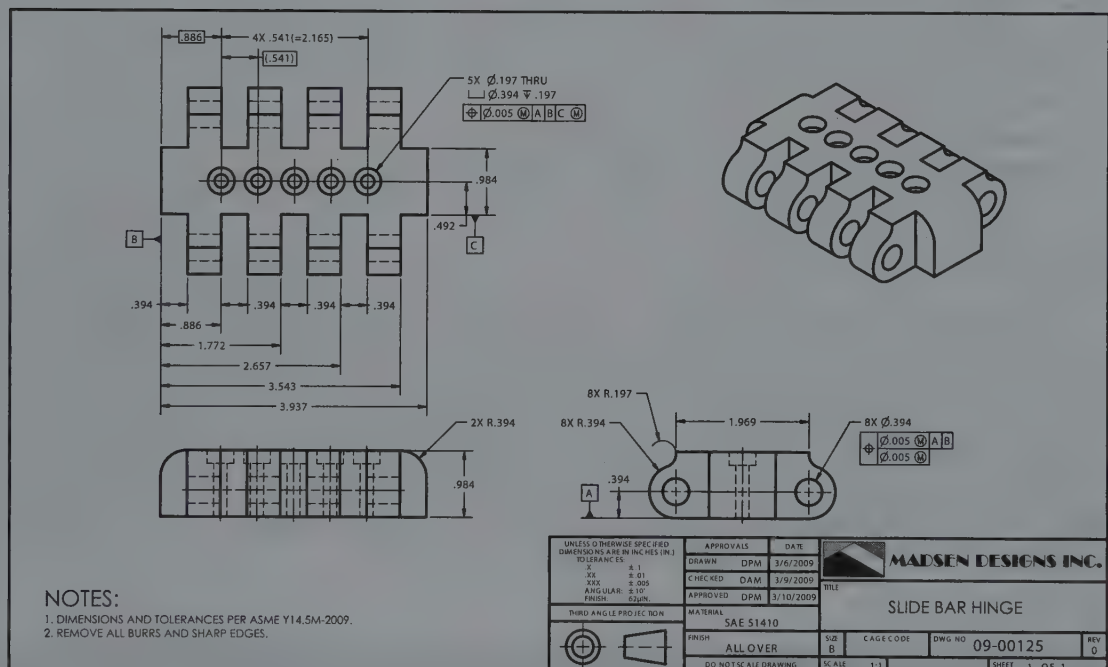
2D Drawing

Exercise 22-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-1.

Drawing Standards

The active drawing standard includes settings and references styles that control the default appearance of drawing views and annotations. For example, layer defaults determine the default color, linetype, line weight, and visibility of objects. Other examples include text styles, which set the default appearance of text, and dimension styles, which control the default display of dimensions. Each style includes multiple

variables. Many styles reference other styles or sub-styles for certain information. For example, a dimension style references a text style for dimension text characteristics. Every drawing object references an object default that establishes the style and layer for the object. For example, center marks reference the **Center Mark** object default, which is assigned a **Center Mark** style and appropriate layer.

The **Style and Standard Editor** is the primary resource for observing and modifying the characteristics of existing standards and styles and for creating and importing new standards and styles. See **Figure 22-2**. Review the **Style and Standard Editor** to become familiar with the vast number of options. However, creating your own standards is a complex and time-consuming process. As a result, this textbook provides predefined template files with standards set according to the American Society of Mechanical Engineers (ASME) national standards. These templates allow you to start preparing drawings immediately, using appropriate drafting standards.

Ribbon

**Manage
> Styles and
Standards**

Styles Editor
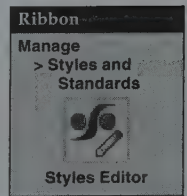
> **NOTE**
>
> A proper standard automates the drawing process by presetting how objects should appear on drawings. However, you can apply overrides when necessary to adjust the display of specific items. An example of a common override is changing the default precision of a dimension to specify a different tolerance.

Figure 22-2.
The **Style and Standard Editor** is the primary tool for controlling the drawing standard and the styles that the standard references.

Styles referenced by the
standard and objects
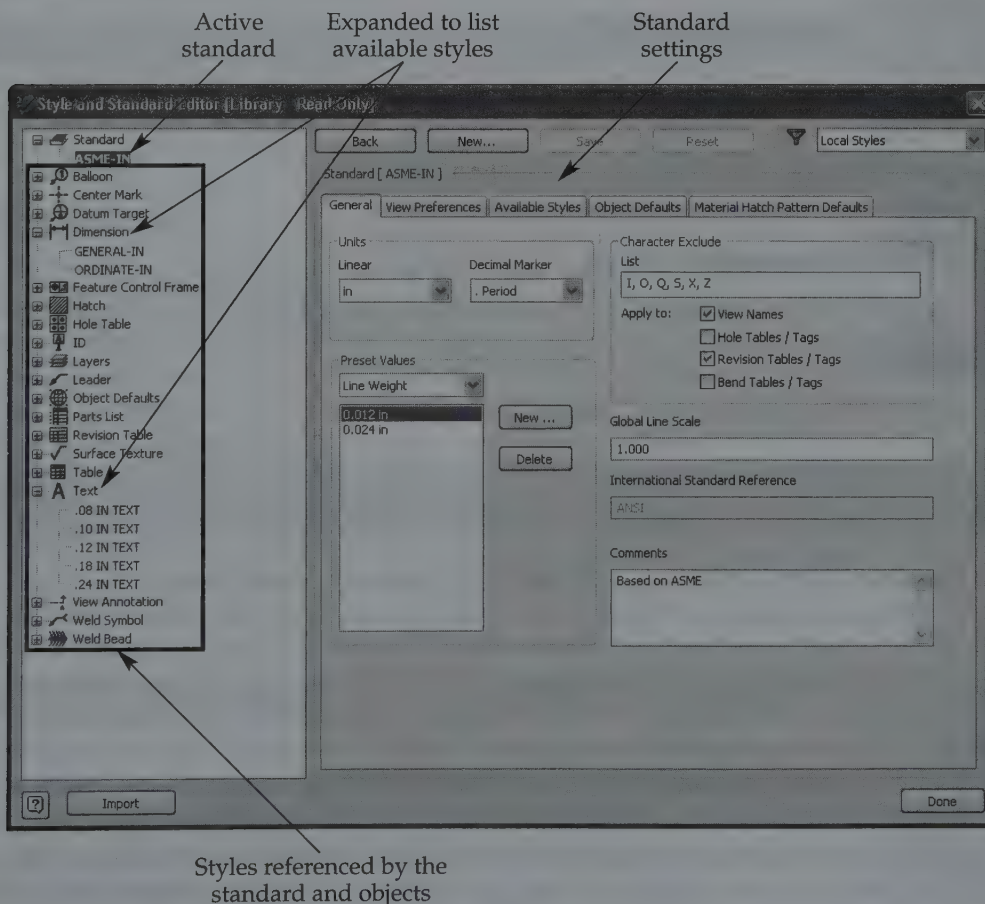
Provided Templates

Drawing templates should include appropriate standards, borders, title blocks, general notes, and other drawing elements that remain constant or require slight modification from one drawing to another. Preparing your own drawing templates often requires significant time and effort because of the multitude of standard and style settings and the complexity of borders and title blocks. As a result, this textbook provides predefined drawing templates that comply with the following standards:

- ASME Y14.1 *Decimal Inch Drawing Sheet Size and Format*
- ASME Y14.1M *Metric Drawing Sheet Size and Format*
- ASME Y14.2M *Line Conventions and Lettering*
- ASME Y14.3M *Multiview and Sectional View Drawings*
- ASME Y14.5M *Dimensioning and Tolerancing*
- ASME Y14.35M *Revision of Engineering Drawings and Associated Documents*

When you enter the drafting and design workforce, you will use existing company templates, or you may need to create or modify templates. Research and explore the modification of standards and other template components to meet your own requirements.

> **NOTE**
>
> Options in the **Drawing** tab of the **Application Options** dialog box also affect drawing appearance and the function of drawing standards. The default settings are appropriate for most applications.

Exercise 22-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-2.

Exercise 22-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-3.

Sheet Preparation

A drawing sheet represents the piece of paper, or sheet, that displays all the drawing content, including a border, title block, views, and annotations. A drawing file can include multiple sheets, which provides the option to group related drawings in a single file. For example, you can prepare a file with an assembly drawing and all associated component drawings. A single sheet named SHEET:1 is available in the Drawing-IN.idw and Drawing-mm.idw templates.

To confirm sheet parameters, or to edit a sheet in preparation for drawing, right-click a sheet in the browser and select **Edit Sheet...** to access the **Edit Sheet** dialog box. See **Figure 22-3**. You can use the **Name** text box to rename the sheet to something more descriptive, such as a part number or the title of the drawing. A number follows the sheet name of included sheets, according to the order of sheets in the drawing. Use the **Size** drop-down list to select the sheet size. To define your own sheet size, select

Figure 22-3.
Use the **Edit Sheet** dialog box to select a sheet size and adjust other sheet parameters.

Enter an appropriate name for the sheet

Specify the sheet size

Set the paper orientation and the location of the title block
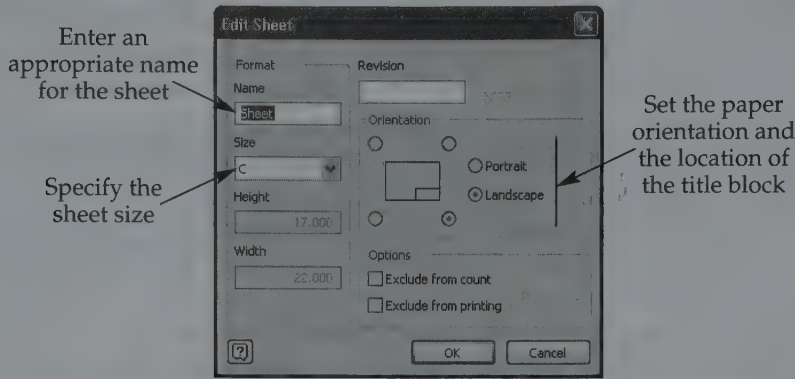
Custom Size (inches) or **Custom Size (mm)** and then specify the height and width using the **Height** and **Width** text box.

Use the **Revision** text box to specify the drawing revision number or level. Pick the **Portrait** radio button to apply a portrait, or vertical, sheet orientation. Select the **Landscape** radio button to apply a landscape, or horizontal, sheet orientation. Title block orientation radio buttons are available after you insert a title block, and allow you to override the default location of the title block in reference to the border. Check **Exclude from count** to exclude the sheet from the count of sheets in the drawing, eliminating the sheet number. Check **Exclude from printing** if you do not want to print or plot the sheet with other sheets. Pick the **OK** button to complete the sheet edit.

> **NOTE**
>
> The **Sheet** tab in the **Document Settings** dialog box provides options for adjusting the default sheet name and sheet display colors.

Adding a Border

A border is sketch geometry that has been saved as a border definition and stored in the **Borders** folder of the **Drawing Resources** folder in the browser. See **Figure 22-4**. The Drawing-IN.idw and Drawing-mm.idw templates include ASME standard borders, specific to sheet size, appropriate for use throughout this textbook. You can insert a border found in the **Borders** folder by double-clicking on the border, or except for the Default Border, by right-clicking on the border and selecting **Insert**.

The border adjusts when you make changes to the sheet size. However, if you use a border specific to a sheet size, the adjusted border will probably be incorrect. Before you can insert a different border, you must delete the existing border. To delete the border, select the border and press [Delete], or right-click on the border and select **Delete**.

> **NOTE**
>
> Inventor includes **Border** and **Zone Border** tools for creating new borders. The **Zone Border** tool automates the process of creating a border with zoning. Borders are sketch geometry created using sketch tools, most of which are identical to those available for sketching model geometry.

Ribbon
Manage
> Define

Border

Shortcut
Define New
Border...

Ribbon
Manage
> Define

Zone Border

Shortcut
Define New
Zone Border...
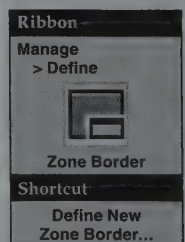
Adding a Title Block

A title block is sketch geometry saved as a title block definition and stored in the **Title Blocks** folder of the **Drawing Resources** folder in the browser. See **Figure 22-4**. The title block contains information about the model, company, drafter, tolerances, and other design and documentation information. The Drawing-IN.idw and Drawing-mm.idw templates include a unit-specific ASME standard title block appropriate for use throughout this textbook. Much of the text in the provided title blocks uses property fields to extract properties from the model, drawing, or sheet.

Model property fields reference entries in the **iProperties** dialog box associated with the base view model. This is appropriate for properties such as title and part number that are typically the same in the model and drawing. Drawing property fields reference the values you enter in the **iProperties** dialog box for the drawing. Sheet properties are items specified in the drawing browser, such as sheet size and number and the total number of sheets. The provided templates also include prompted entry fields that allow you to enter values that are not extracted from a model, drawing, or sheet. Respond to prompted entries when you insert the title block using the **Prompted Text** dialog box, shown in **Figure 22-5A**, or edit the values later.

You can insert a title block found in the **Title Blocks** folder by double-clicking on the title block or by right-clicking on the title block and selecting **Insert.** The title block is automatically placed at the specified corner of the border. To edit property field text, access the source of the properties, such as the **iProperties** dialog box in the model or drawing. To edit prompted or custom field text, double-click on **Field Text** consumed by the title block in the browser, or right-click and select **Edit Field Text...**, to access the **Edit Property Fields** dialog box. See **Figure 22-5B**. To delete the title block, as necessary to insert a different title block, select the title block and press [Delete], or right-click on the title block and select **Delete**.

Figure 22-4.
The provided Drawing-IN.idw and Drawing-mm.idw templates include borders, title blocks, and general notes appropriate for most drawing applications using ASME standards. This figure shows using the Drawing-IN.idw template.
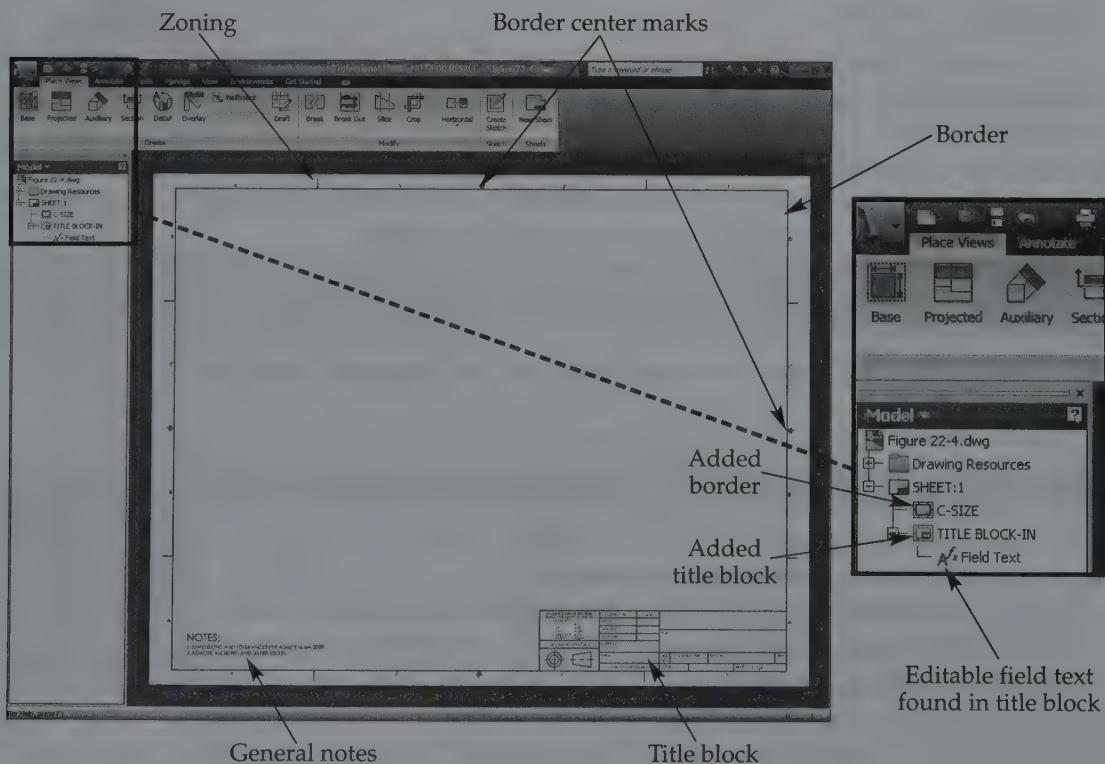
Figure 22-5.
A—The **Prompted Text** dialog box appears when you insert a title block that includes prompted text fields. B—Edit prompted or custom text fields using the **Edit Property Fields** dialog box. You must use fields extracted from model or drawing iProperties in the model or the drawing **iProperties** dialog box.

Enter a value

Enter a value

A   B
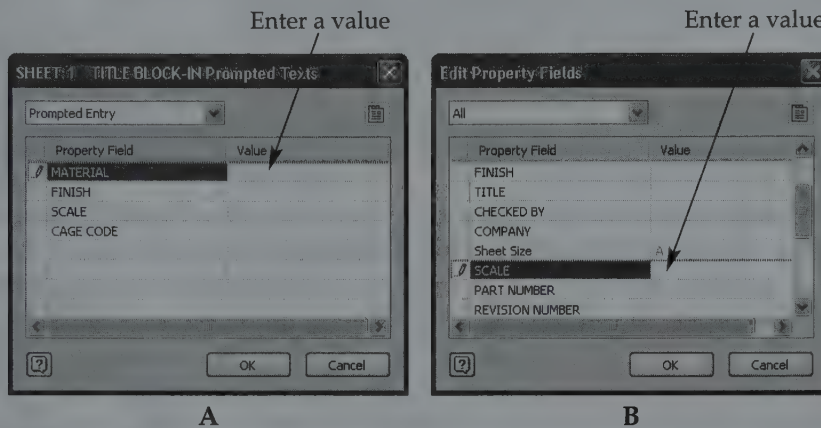
PROFESSIONAL TIP

Model, drawing, and sheet property field text links the drawing to the corresponding model and sheet. If you correctly assign iProperties to a model and arrange sheets in the correct order in the drawing browser, title block cells should fill automatically when you create sheets and insert a base view.

NOTE

Inventor includes a **Title Block** tool for creating new title blocks. Title blocks are sketch geometry created using sketch tools, most of which are identical to those available when sketching model geometry.

Ribbon
Manage
> Define
Title Block
Shortcut
Define New Title Block

General Notes

The Drawing-IN.idw and Drawing-mm.idw templates include two *general notes* that are common to most drawings. Refer again to **Figure 22-4**. The notes are created using the **Text** tool, allows you to save the notes with the template and edit the list of notes as needed. However, positioning text created using the **Text** tool is not ideal. You are responsible for dragging the notes to approximately .5″ (13 mm) away from the border or title block. Chapter 23 covers using the **Text** tool.
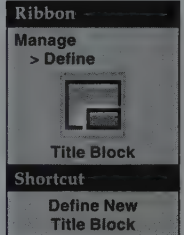
general notes: Notes that apply to the entire drawing, usually placed together in a standard location, such as the lower-left or upper-right corner of the border or near the title block.

NOTE

In the drawing environment, use view, measure, and other common tools and options just as you would use the tools and options in a model, but remember that you are working in a 2D environment.

Ribbon
Annotate
> Text
A
Text
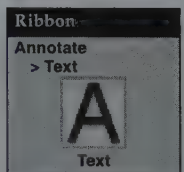
Exercise 22-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-4.

Exercise 22-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-5.

Base Views
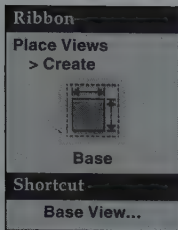
base view: The initial drawing view, from which you can create other views if necessary.

Access the **Base View** tool to create a *base view* using the **Drawing View** dialog box. See **Figure 22-6**. Use the options in the **Component** tab, shown in **Figure 22-6**, to select a model file to reference to create the view. You can select a currently open file from the **File** drop-down list. To reference a model that is not open, pick the **Browse** button to locate and select the file. The file should appear in the **File** drop-down list.

The content of the **Component** tab varies depending on the model you select. When you select a sheet metal part, the **Sheet Metal View** area appears, allowing you to pick the appropriate radio button to create a folded part or flat pattern view. See **Figure 22-7**. Sheet metal part drawings sometimes show folded part views and a flat pattern. See **Figure 22-8**. This technique requires using the **Base View** tool twice to add a folded and flat view. Apply a similar technique to create a multidetail drawing. The **Component** tab includes additional options when placing a view of an assembly, weldment, or presentation model.

Options in the **Model State** tab become enabled when you place an assembly or weldment view, as described in Chapter 24, or when you reference an iPart or iAssembly. The **Display Options** tab, shown in **Figure 22-9**, contains multiple check boxes corresponding to items that may be found in the selected model, and that you can display in

Figure 22-6.
Using the **Drawing View** dialog box to place the initial drawing view, a front view.
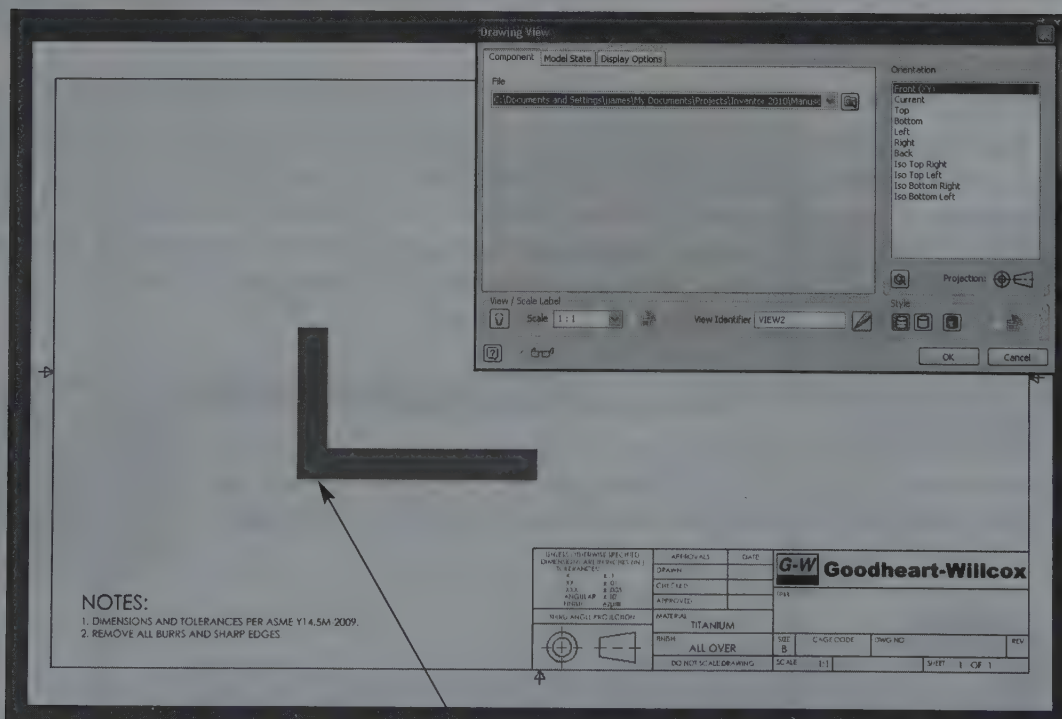
Pick to locate the view

Figure 22-7.
The **Sheet Metal View** area on the **Component** tab of the **Drawing View** dialog box.
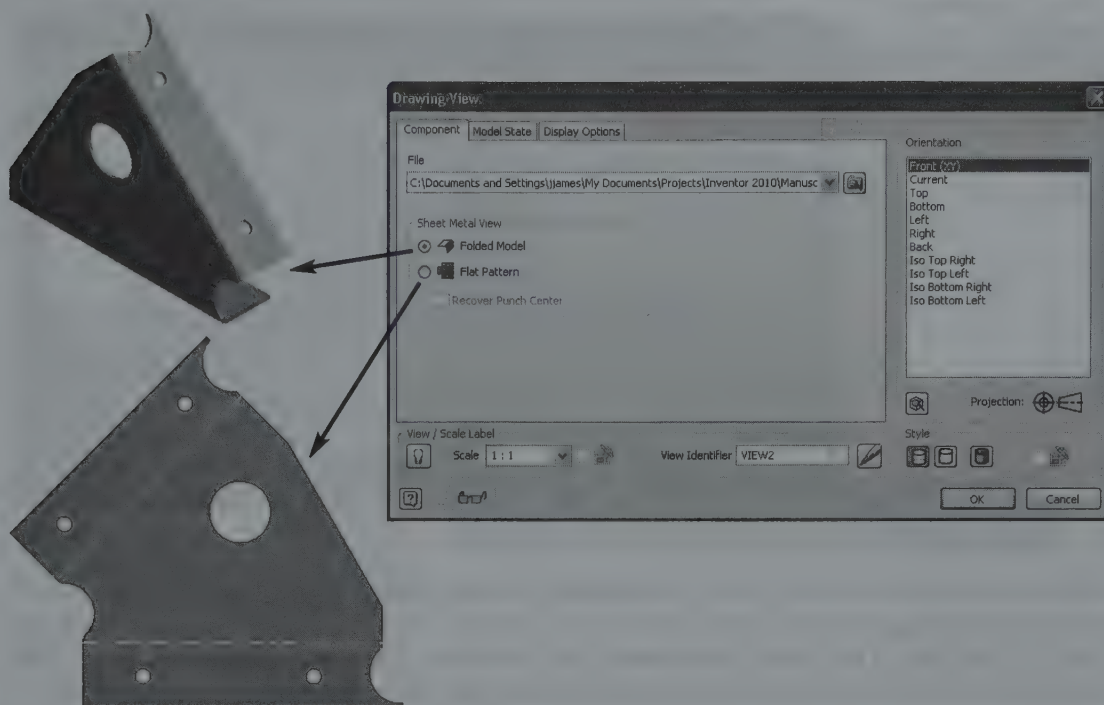
Figure 22-8.
An example of a drawing that requires two base views to show a folded part and a flat pattern. Additional views are projected from the folded part base view.

Folded part
base view

Flat pattern
base view
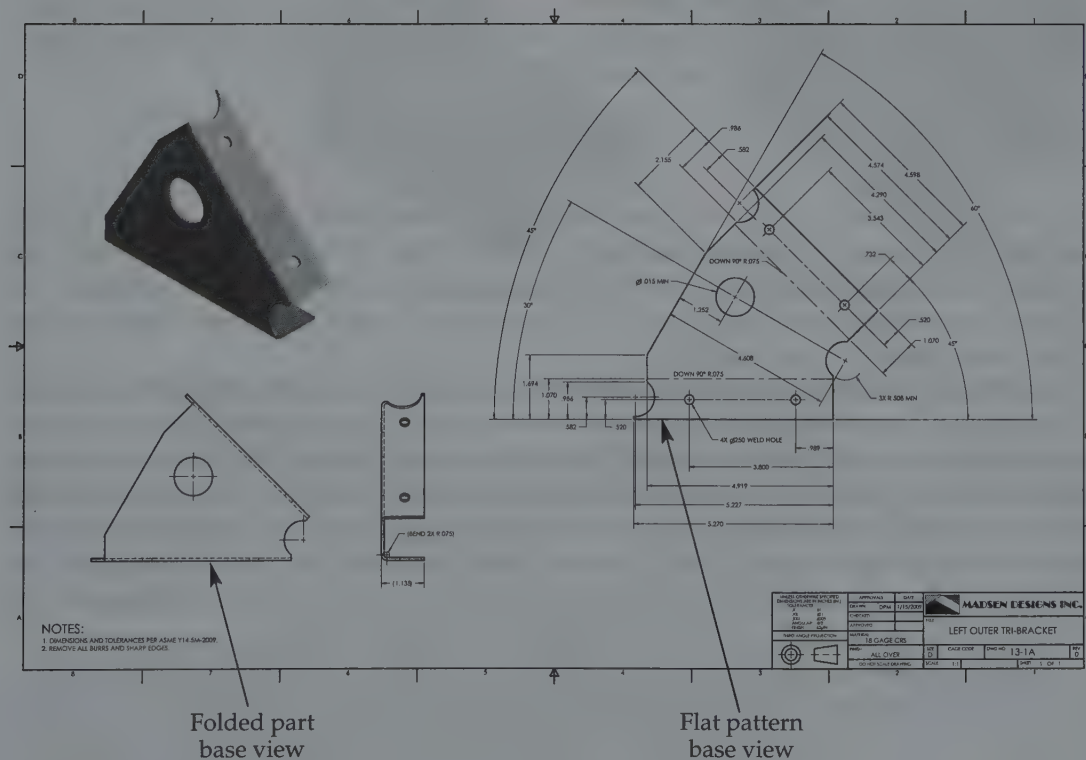
Figure 22-9.
The **Display Options** tab of the **Drawing View** dialog box.

Pick to display model
dimensions in the drawing
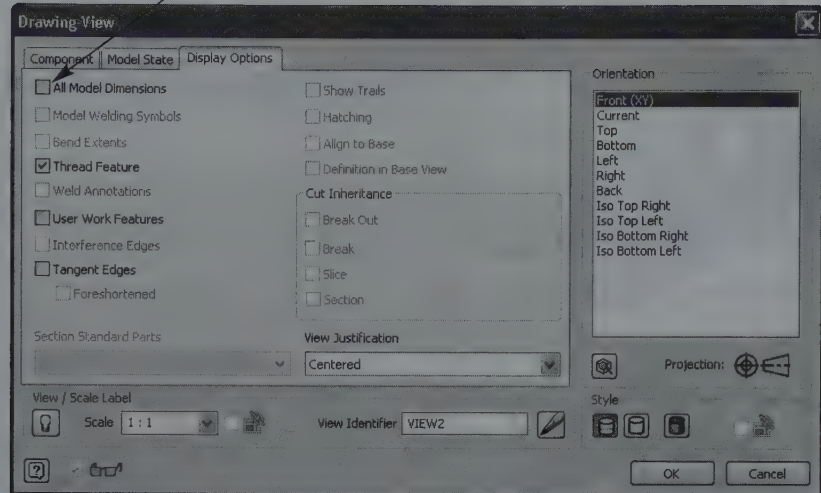
the base view. For example, check **All Model Dimensions** to extract and display model dimensions planar to the base view. If a check box is disabled, the model does not include the item. The **View Justification** and **Section Standard Parts** drop-down lists allow you to override the default position of the base view and standard part sectioning in the base view, as specified in the **Drawing** tab of the **Options** dialog box.

> **NOTE**
>
> The **Foreshortened** check box becomes enabled when you check **Tangent Edges**, allowing you to shorten tangent edges to differentiate them from other objects. Displaying tangent edges is not common drafting practice.

multiview drawing: 2D views of an object projected onto planes of projection; represents the shape of the object in two or more views.

front view: The view that provides the best shape description and most characteristic contours, has the longest dimension, has the fewest hidden features, and is the most stable and natural position.

Ribbon
Custom View > Appearance
Orthographic

Ribbon
Custom View > Appearance
Perspective

Orientation

The **Orientation** area allows you to select the base view orientation, or projection. Several common views are available, such as **Front (XY)** and **Top**. By default, the preview of the view in the graphics window changes to reflect the view selection. For a typical *multiview drawing*, display the *front view* as the base view. You can then project additional views as needed.

Develop models according to drawing views when possible. This means that when you pick the **Front (XY)** orientation option, for example, the correct front view of the model appears in the drawing. However, creating a model that corresponds to specific drawing views can be difficult, especially for complex or significantly redesigned models. As a result, you may have to select an alternative base view orientation in order to achieve the correct drawing view. For example, pick the **Right** or **Top** orientation option to create the correct front view. The alternative is to redefine model sketches or recreate model geometry, which can be time-consuming.

If none of the available views are appropriate, pick the **Custom View** button to open the **Custom View** window, shown in **Figure 22-10**. Standard viewing tools are available to adjust the model view as needed. The most common application involves using tools such as **View Cube**, **Orbit**, or **View Face** to display a specific orthographic view. You also have the option of changing the projection mode from **Orthographic** to **Perspective** to create a perspective view. See **Figure 22-11**.

Figure 22-10.
The **Custom View** window provides view tools for creating a view when none of the default views is acceptable.

Custom view tools
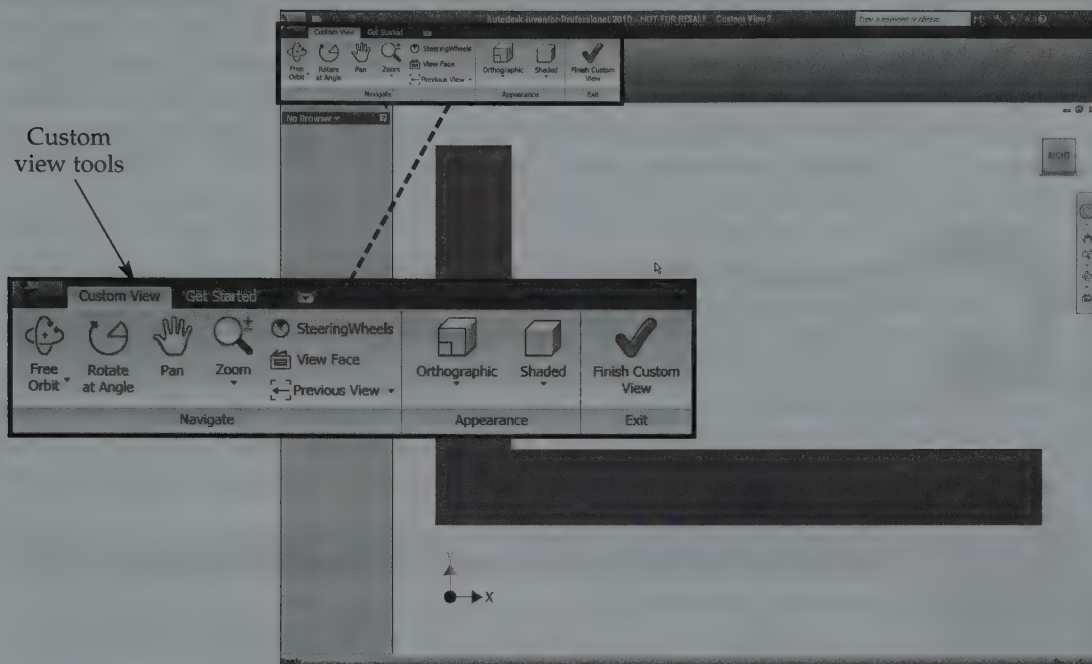
Figure 22-11.
Creating a perspective view using rotate tools and the **Perspective** view mode.

Orthographic   Perspective
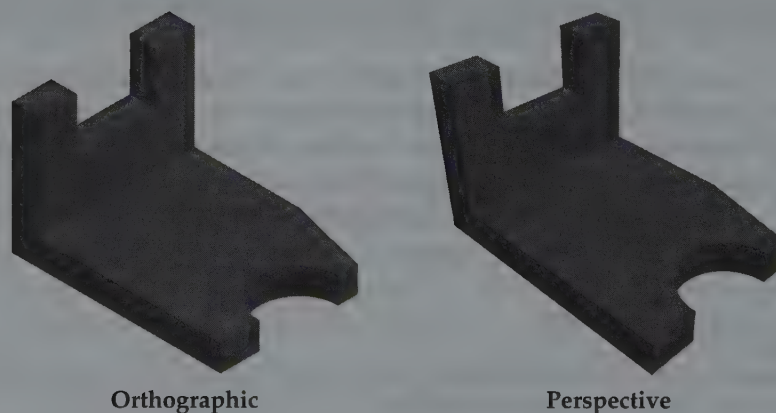
The **Rotate at Angle** tool allows you to use the **Incremental View Rotate** dialog box to orbit the view by an incremental value, such as 10° clockwise. This is the same dialog box that is available when you use the **Precise View Rotation** tool in the presentation work environment. When you are satisfied with the custom view display, pick the **Finish Custom View** button on the **Exit** panel of the **Custom View** ribbon tab.
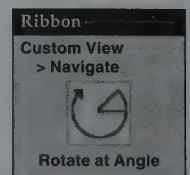
Scale and Label

Select a suitable view scale from the **Scale** flyout or enter a different scale in the text box. The **Scale from Base** check box is unavailable because a base view is not *dependent* on another view. Use the **View Identifier** text box to change the view name that appears in the browser and with the view label, if used. For most views, the default name is acceptable. Use the **Toggle Label Visibility** button to display the view name and scale with the view, but this practice is not necessary for most views. If you do require the label, you can pick the **Edit View Label** button to make changes to the default label using the **Format Text** dialog box.

dependent view:
A view projected from and linked to another view, such as a base view.

Style and Finalizing

Use the **Style** area to specify the style of view geometry. Pick the **Hidden Line** button to display the view with hidden lines, which is suitable for typical multiview drawing. Pick the **Hidden Line Removed** button to remove hidden lines, or select the **Shaded** button to shade the view. These options are most appropriate when you are creating an isometric or similar view. The **Style from Base** check box is unavailable because the base view is not dependent on another view. To place the base view, pick a location in the graphics window or select the **OK** button.

Exercise 22-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-6.

Exercise 22-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-7.

Projected Views

orthographic view: A 2D view, or projection, in which the line of sight is perpendicular to a surface, such as the front of an object or the XY plane.

isometric view: A 3D view in which all three axes appear at equal (120°) angles with the plane of projection.

Access the **Projected View** tool to create standard *orthographic* or *isometric* projections from a base or dependent view. Select a view if you have not already, and move the cursor to preview projection options. When you see the correct projection in the appropriate location, pick to locate the view. Continue picking projected view locations, including isometric views, as needed. When you are satisfied with the number of projections, right-click and select **Create** to generate the views and exit. See **Figure 22-12**.

Figure 22-12.
You can use the **Projected View** tool to place all additional standard orthographic and isometric views in a single operation.

Ribbon
Place Views > Create

Projected

Shortcut
Create View > Projected View...

Picking Views   Final Projections
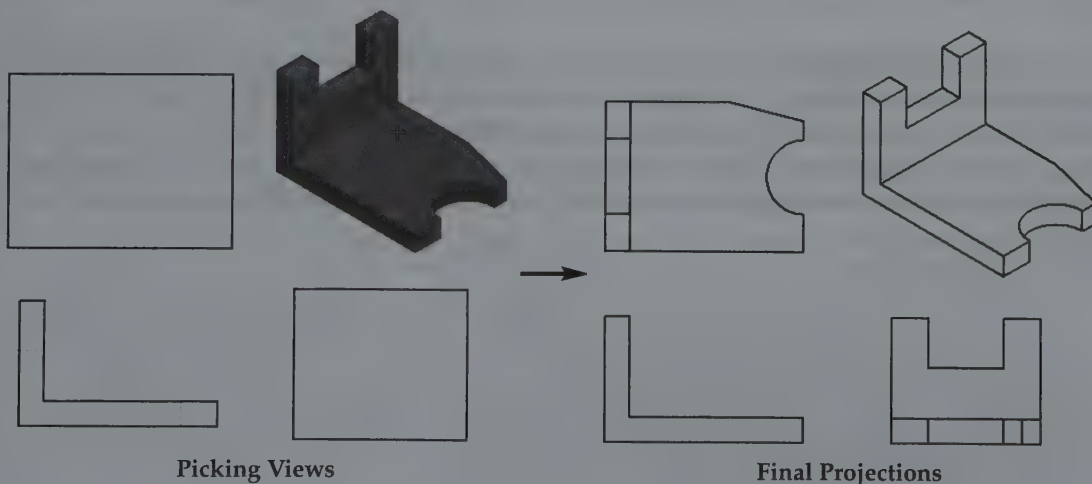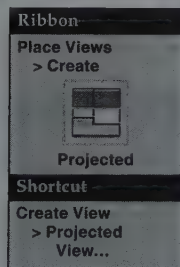
Often you can place all additional orthographic and isometric views using a single **Projected View** tool operation. However, you can use the **Projected View** tool whenever you need it throughout the drawing process.

Basic View Adjustment

Drawing views reference model geometry, the active drawing standard, and settings you specify when you create the views. You can open a model to make changes from inside the drawing by right-clicking on a view in the browser or graphics window and selecting **Open**. To make a change to a view using options in the **Drawing View** dialog box, right-click on a view in the browser or graphics window and select **Edit View....** This allows you to change base view parameters, but is also commonly used to adjust views that were not placed using the **Drawing View** dialog box, such as projected views. Use the dialog box to make changes such as choosing a different style or controlling display options. For example, you can assign a **Shaded** style to a projected isometric view.

You can view and adjust basic properties of model content from within a drawing by expanding a view to display the model. Right-click on model features and pick **Properties...** to access basic properties.

Moving Views

Moving views is often necessary to create correct drawing layout and space between views. Base views are free to move anywhere in 2D space, but by default, you have limited ability to move other views, according to the link between views or view geometry. To move a view, hover the cursor over the view boundary until you see the view boundary cursor. Then hold down the left mouse button, drag the view to the desired location, and release the left mouse button. See **Figure 22-13**.

Figure 22-13.
Moving a drawing view.

View alignment lines

Hold down and drag to move view
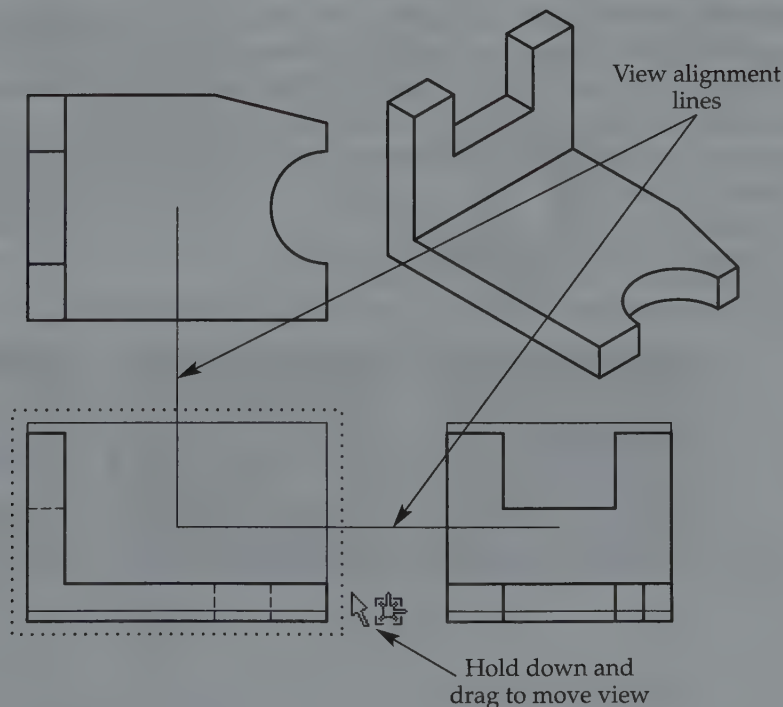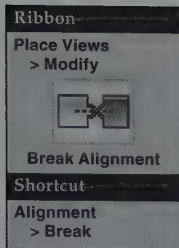
View Alignment

Alignment constraints form by default between most parent views and dependent views, limiting the movement of dependent views. This is appropriate for most applications to maintain correct alignment between views. For example, orthographic projections remain horizontal or vertical to a base view even if you move views. When direct alignment is not possible because of view layout or sheet requirements, you can create a removed view using the **Break Alignment** tool. Inventor automatically creates a viewing-plane line and labels the broken alignment view. Right-click on the label and select **Edit View Label** to edit the label using the **Format Text** dialog box. For example, you might need to remove the scale reference.
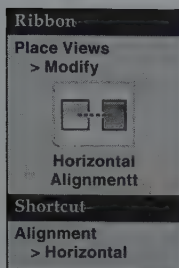
You have the option of establishing alignment if no alignment exists, or when correct alignment does not apply. If alignment constraints already exist, you must first break the alignment as described. Then establish new alignment between views, one of which is usually a base or primary view. Use the **Horizontal Alignment** tool to horizontally align two selected views. Access the **Vertical Alignment** tool to vertically align two selected views. The **In Position** alignment tool allows you to create an alignment constraint that is neither vertical nor horizontal, which is the alignment applied by default to isometric view projections
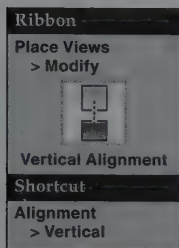
Deleting Views

To delete a view, pick the view in the browser or the view boundary in the graphics window and press [Delete], or right-click on the view and select **Delete**. An alert box confirms the deletion. If you attempt to delete a view that other views reference, such as a base view, or depend on, such as projected views, the alert box allows you to choose whether dependent views are also deleted. Pick the **More** button to display a list of dependent views. The default Yes option deletes dependent views. Pick Yes and it changes to No, preventing removal of dependent views.
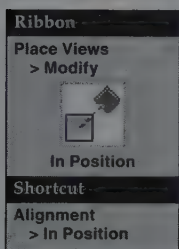
Rotating Views

Right-click on a view in the browser or graphics window and select **Rotate** to rotate a view using the **Rotate View** dialog box. See **Figure 22-14**. The **Edge** option, available from the drop-down list, rotates the view by aligning a selected edge horizontally or vertically according to the selected **Horizontal** or **Vertical** radio button. The **Absolute Angle** and **Relative Angle** options, available from the drop-down list, rotate the view to an angle specified in the **Angle** text box. The **Absolute Angle** option rotates the view according to the sheet's coordinate system. The **Relative Angle** option rotates the view from the current orientation of the view. Use the **Counterclockwise** button to rotate the view counterclockwise, or the **Clockwise** button to rotate the view clockwise. Pick the **OK** button to exit.

Figure 22-14.
Use the **Rotate View** dialog box to rotate a view using an edge or an angle.
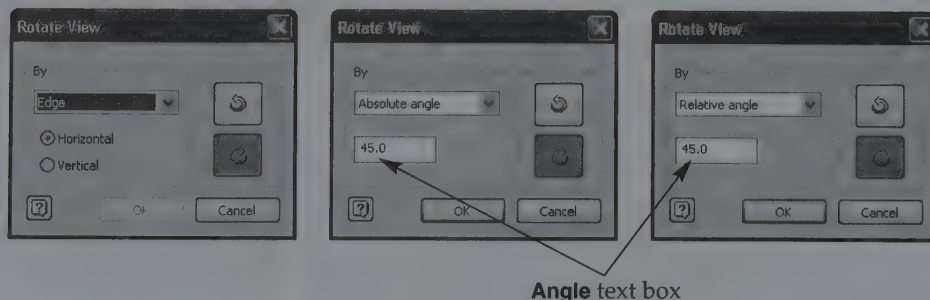
Angle text box

Selecting Objects

Drawing requires that you select objects, much like the selections required while sketching and modeling. For example, you must select objects to dimension, or you may want to turn off object visibility. Select single objects using the cursor. Use the **Select Other** tool to cycle through and select the appropriate object. Hover the cursor over objects until you see the **Select Other** tool. To select multiple objects, hold down [Ctrl] or use window or crossing selection.

Exercise 22-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-8.

Auxiliary Views

auxiliary view: A view that shows the true size and shape of an inclined surface that is not parallel to any of the projected views, including the front, top, bottom, left-side, right-side, and back views.

Access the **Auxiliary View** tool to project an *auxiliary view* from an existing view onto the auxiliary plane. Select a view, if you have not already, to display the **Auxiliary View** dialog box. See **Figure 22-15**. Specify the view identifier and scale using the corresponding text boxes. Use the **Toggle Label Visibility** button to display the view label, and use the **Edit View Label** button to make changes to the default label. Specify the style of view geometry in the **Style** area.

Ribbon
Place Views
> Create

Auxiliary

Shortcut
Create View
> Auxiliary View...

Figure 22-15.
Using the **Auxiliary View** tool to create an auxiliary view.

Toggle Label Visibility button

Auxiliary view preview

Projection line

Selected edge to project auxiliary view

Selected view (base view)

Projected right-side view
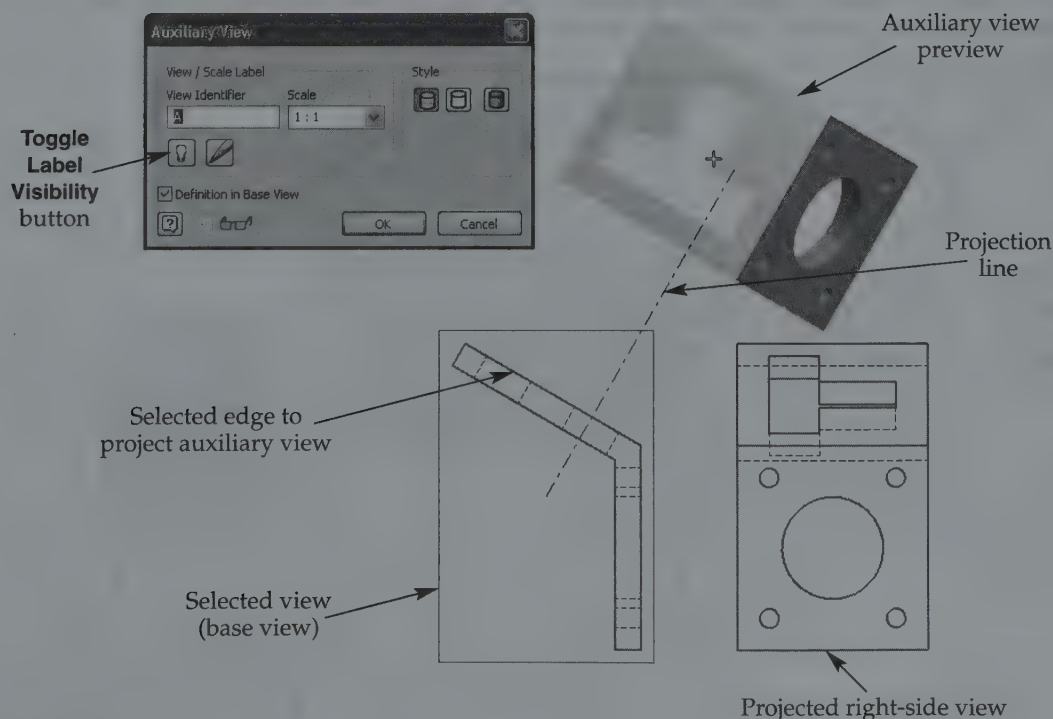
Check **Definition in Base View** only if you want to add a viewing-plane line. Typically, a viewing-plane line is necessary only when you break alignment and remove the auxiliary view, as shown in **Figure 22-16**. Deselect the **Definition in Base View** check box for most applications. Inventor automatically creates a viewing plane line and labels the broken alignment view. Next, pick the edge of the surface from which the auxiliary view projects, and move the cursor to preview the auxiliary view. When you see the correct projection in the appropriate location, pick in the graphics window or select the **OK** button to place the auxiliary view.

> **NOTE**
>
> You can redefine the edge from which the auxiliary view is projected without deleting and recreating the auxiliary view. Right-click on the auxiliary view in the browser or graphics window and select **Realign auxiliary views**. Pick an edge, and then pick the location of the auxiliary view.

Partial Auxiliary Views

partial auxiliary view: A view that shows the true size and shape of only the inclined surface, eliminating any projected geometry that may be foreshortened.

There are no tools that allow you to create a *partial auxiliary view* directly, unless you create an auxiliary view section. To create a partial auxiliary view, right-click on the objects to hide and deselect **Visibility**. See **Figure 22-16A**. Turn off the visibility of all foreshortened and hidden geometry until you achieve the correct partial auxiliary view. See **Figure 22-16B**.

> **NOTE**
>
> You can turn off the visibility of most objects in all types of views, not just auxiliary views.

Figure 22-16.
A—An example of breaking the alignment between a base view and an auxiliary view when space is limited. Inventor automatically places a viewing-plane line and label as required. Edit the label to remove the scale reference. B—An example of a partial auxiliary view created by turning off the visibility of foreshortened objects.
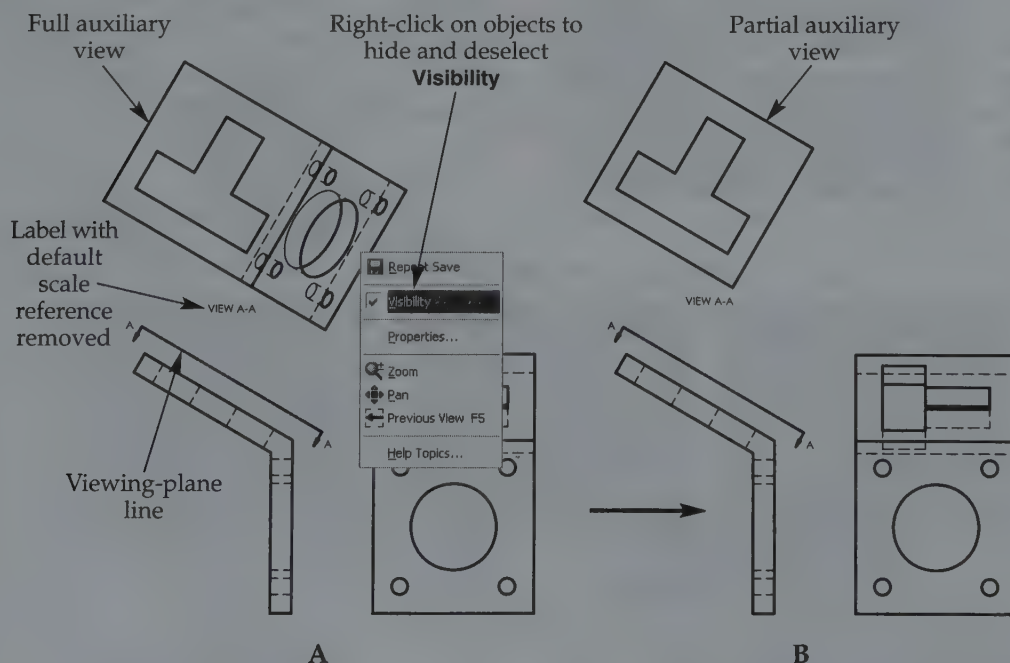
Exercise 22-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-9.

Section Views

Use the **Section View** tool to create a *section view* projected from a *cutting plane* through an existing view. Select a view if you have not already, and then pick points to construct a *cutting-plane line*. The technique for creating a cutting-plane line is similar to sketching a line. When you are finished, right-click and select **Continue** to display the **Section View** dialog box. See **Figure 22-17**.

section view (sectional view, section): A view that exposes the interior features of a component, typically eliminating hidden lines and allowing you to dimension geometry and provide a clearer representation.

cutting plane: An imaginary plane along which the cut is made to create a section.

cutting-plane line: The line that represents the cutting plane of a section.

Figure 22-17.
A—Select the view from which to project the section view, and then pick points to construct the cutting-plane line (full section shown). Infer geometric constraints and use point alignment to locate points. B—Right-click and select **Continue** to finish the cutting-plane line. C—Use the **Section View** tool to finalize a full section. D—A full section projected from a top base view. Edit the label to remove the scale reference.

You can also use sketch tools to sketch a cutting-plane line *before* accessing the **Section View** tool. This method offers additional options for cutting-plane line placement. You must associate the sketch with the view by picking the view before accessing the **Create Sketch** tool. Finish the sketch, access the **Section View** tool, and pick the sketch as the cutting-plane line.

full section:
A section that shows the entire plane inside a part, created from a single cutting-plane line placed completely through the view.

half section:
A section used when an area of a part does not require sectioning because the area has no features, or because the features are symmetrical to the sectioned features.

offset section:
A section from a cutting-plane line that travels through various depths of a part when the features cannot be seen with a straight cutting-plane line.

aligned section:
A section from a cutting-plane line that changes directions at an angle other than 90° when the cutting-plane line must go through features that revolve around the center of a cylindrical part.

Figure 22-17 shows an example of creating a *full section*. **Figure 22-18** shows examples of creating a *half section*, an *offset section*, and an *aligned section*. Notice the correct position of the parent views and where you place the cutting-plane line, shown in **Figure 22-17** and **Figure 22-18**. When creating an aligned section, you must use the **Aligned** method, as explained later in this chapter.

Specify the view identifier and scale using the corresponding text boxes. Use the **Toggle Label Visibility** button to display the view label and the **Edit View Label** button to make changes to the default label. Use the **Style** area to specify the style of view geometry. Pick the **Full** option from the **Section Depth** drop-down list to remove all material from the cutting-plane line out, which is appropriate for most applications. Select the **Distance** option and then specify the portion of material retained using the **Distance** text box. **Figure 22-19** shows an example of each depth option.

Use the **Projected** method to apply direct section view projection from the cutting plane. If direct projection results in undesirable foreshortening, pick the **Aligned** method to create an aligned section. Once you create the cutting-plane line and adjust section view parameters, move the cursor to preview the section view. When you see the correct projection in the appropriate location, pick in the graphics window or select the **OK** button to place the section view.

Figure 22-18.
Examples of creating half, offset, and aligned sections. Notice the **Hidden Lines Removed** style applied to section views.

Half Section
(Projected)

Offset Section
(Projected)

Aligned Section
(Aligned)
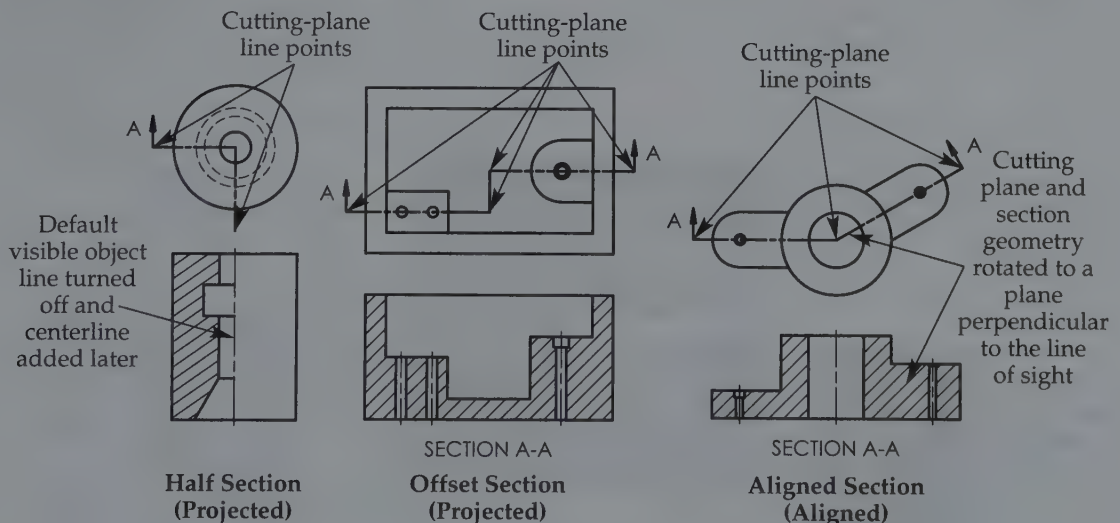
Figure 22-19.
Section views created using the **Full** depth option and the **Distance** depth option.

SECTION A-A
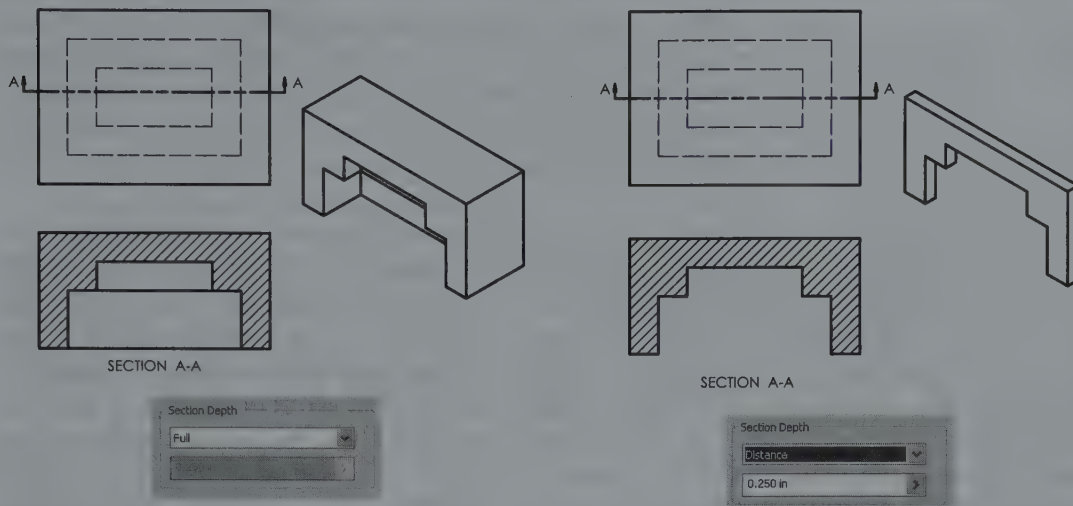
SECTION A-A

Zero-Depth Sections

Pick the **Include Slice** check box in the **Section View** dialog box to create a sliced, or zero-depth, section. This is the same as using the **Distance** depth option and specifying a 0 distance. Inventor also provides a **Slice** tool that allows you to slice a view using a sketch associated with a different view. Zero-depth sections are sometimes suitable for applications that require multiple sections without showing unneeded geometry, such as documenting the contour of an impeller or propeller. See **Figure 22-20**.

Ribbon

**Place Views
> Modify**
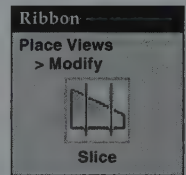
Slice

Figure 22-20.
An example of a zero-depth section application. In this example, the **Slice** tool was used to create the sections.

Selected sketch
(cutting planes)

Zero-depth
sections

View selected
to section
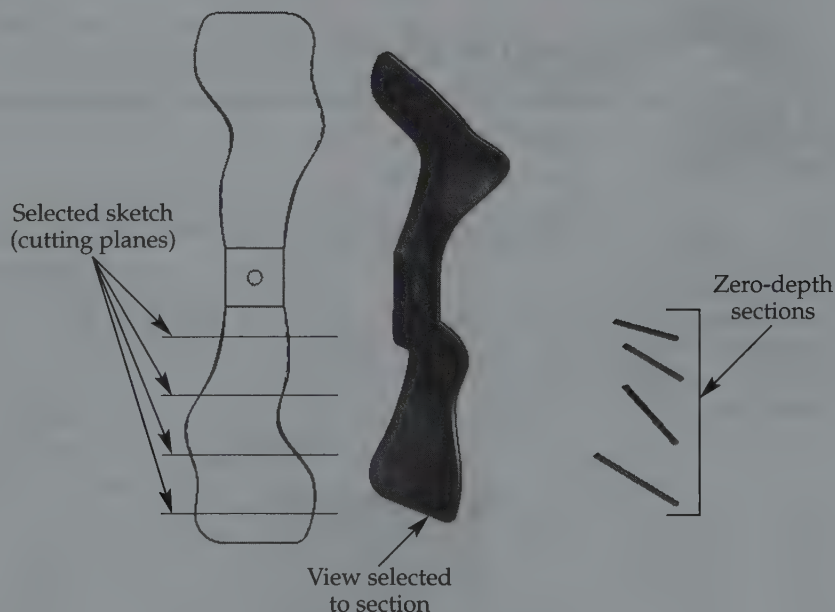
Section View Adjustment

A cutting-plane line, *section lines*, and a label, if set to visible, automatically appear when you create a section view. The default **Section View** dialog box settings create the appropriate view for most applications, but modifications are sometimes required. You have significant control over the appearance of section view elements. For example, to display a correct half section, turn off the visibility of the object line that forms by default, and add a centerline, as described in Chapter 23. Refer again to **Figure 22-18**.

Adjust a cutting-plane line by dragging the grips at the ends of the cutting-plane line that appear when you hover over the line. You can also edit the cutting-plane line in the sketch environment by right-clicking on the cutting-plane line in the browser or graphics window and selecting **Edit**. You can reverse the cutting-plane direction arrows by right-clicking on a cutting-plane line and selecting **Reverse Direction**, although this is usually not appropriate. The default style creates full-length cutting-plane lines. To remove the inner portion of a cutting-plane line, right-click on the cutting-plane line and deselect **Show Entire Line**. See **Figure 22-21**. To edit the section depth, right-click on a cutting-plane line and pick **Edit Section Depth** to open the **Edit Section Depth** dialog box, which functions the same as the **Section Depth** area of the **Section View** dialog box.

The default section lines are displayed according to the hatch style settings assigned to the active standard. To change the display of section lines, right-click on the section lines and pick **Modify Hatch...** to open the **Modify Hatch Pattern** dialog box. The most common change for a part drawing is to adjust the angle in the **Angle** text box so that section lines are not parallel or perpendicular to objects. Changing the pattern, scale, and shift are most common for sectioning an assembly, to distinguish between different components. To turn off section line visibility, right-click on the section lines and select **Hide Hatch**.

Exercise 22-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-10.

Figure 22-21.
Use the **Show Entire Line** option to override the default full-length cutting-plane line.

Show Entire Line On   Show Entire Line Off
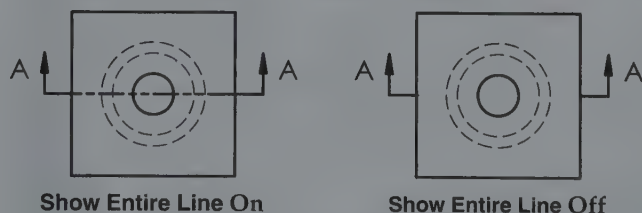
Broken-out Section Views

To create a *broken-out section view,* you first sketch a closed profile that *must be associated with the view,* identifying where to remove material. Pick the view to associate and then access the **Create Sketch** tool. You know the view is active when the view boundary appears and the view is highlighted in the browser. Sketch a closed profile, such as the examples shown in this chapter, and finish the sketch.

Access the **Break Out View** tool and select the view with the associated sketch profile, if you have not already, to display the **Break Out View** dialog box. See **Figure 22-22**. If a sketch includes one profile, the profile is selected automatically. If a sketch contains multiple profiles, the **Profile** button is active, allowing you to pick the profile(s) to break out. The **Depth** area allows you to specify how much of the material to remove. **Figure 22-22** shows using the **From Point** option to specify the depth by picking a point on a view that displays depth. If necessary, you can adjust the depth from the selected point by specifying an offset in the text box. Pick the **OK** button to create the broken-out section.

Ribbon

Place Views
> Modify

Break Out

Shortcut

Create View
> Break Out...

NOTE

The **Show Hidden Edges** button is available with any of the depth options. Pick to have temporary access to hidden lines that may not appear in the view, in order to make selections.

Figure 22-23 shows using the **To Sketch** option to specify the depth by picking a sketch associated with a view that displays depth. This technique requires that you sketch the profile associated with one view, and sketch the depth geometry on the other view, before you access the **Break Out View** tool. Then, with the **Break Out View** tool active, select the profile, pick the **To Sketch** option, and select the depth sketch.

Figure 22-24 shows using the **To Hole** option to specify depth by picking a hole on any view to section to the axis of the hole. **Figure 22-25** shows using the **Through Part** option to section through the thickness of a selected part. Pick the **OK** button to create

Figure 22-22.
Using the **From Point** option of the **Break Out View** tool to create a broken-out section.

From Point

Sketch profile

Sectioned to depth
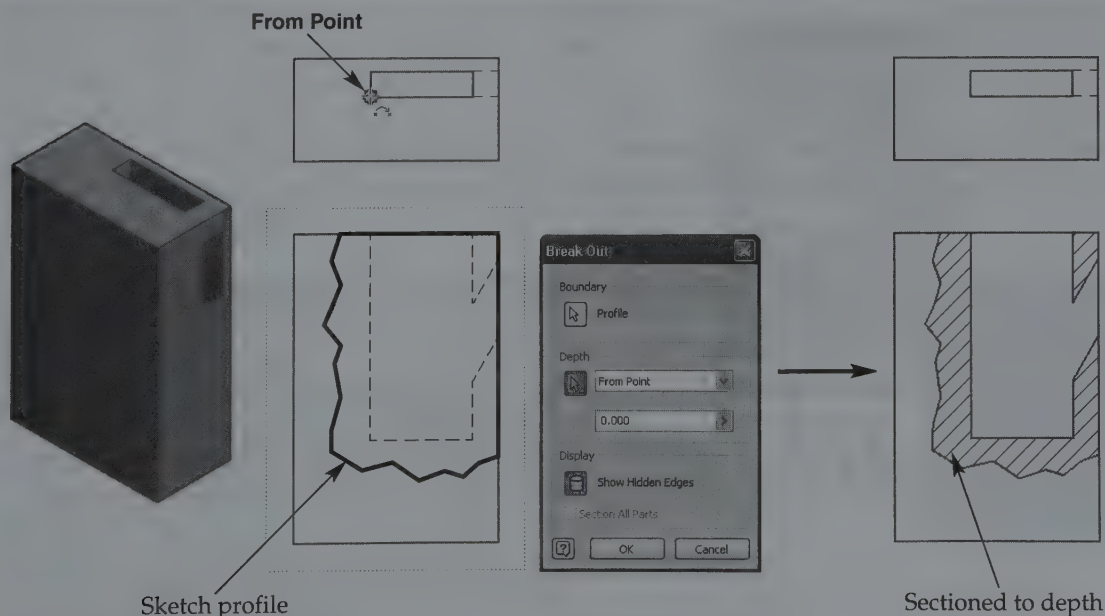
Figure 22-23.
Using the **To Sketch** option of the **Break Out View** tool to create a broken-out section.

Sectioned to depth

Sketch profile   Sketch depth
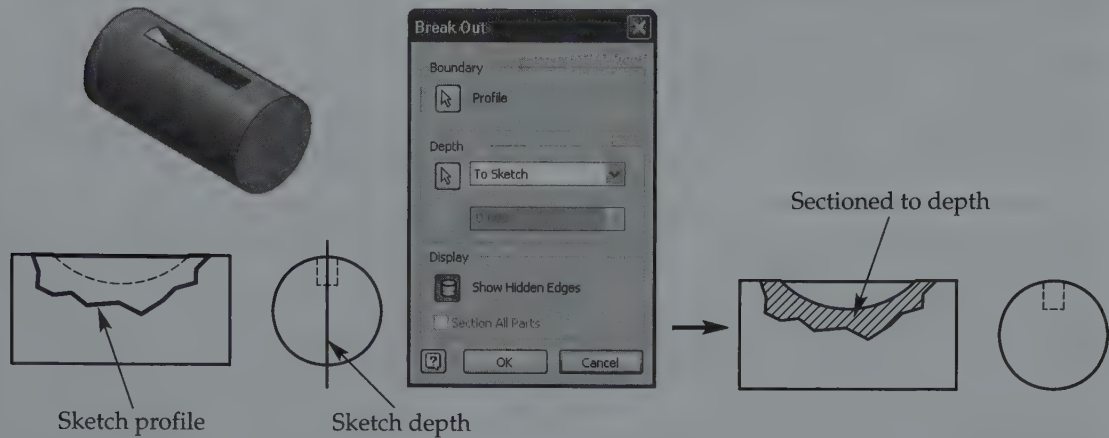
Figure 22-24.
Using the **To Hole** option of the **Break Out View** tool to create a broken-out section.

Hole

Sketch profile   Sectioned to depth
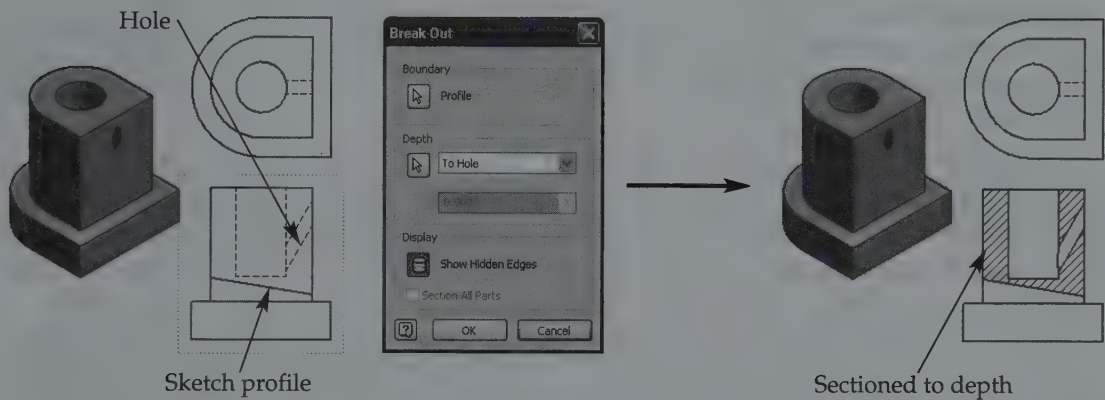
Figure 22-25.
Using the **Through Part** option of the **Break Out View** tool to section through the thickness of a part.

Sectioned through part

Sketch profile   Part
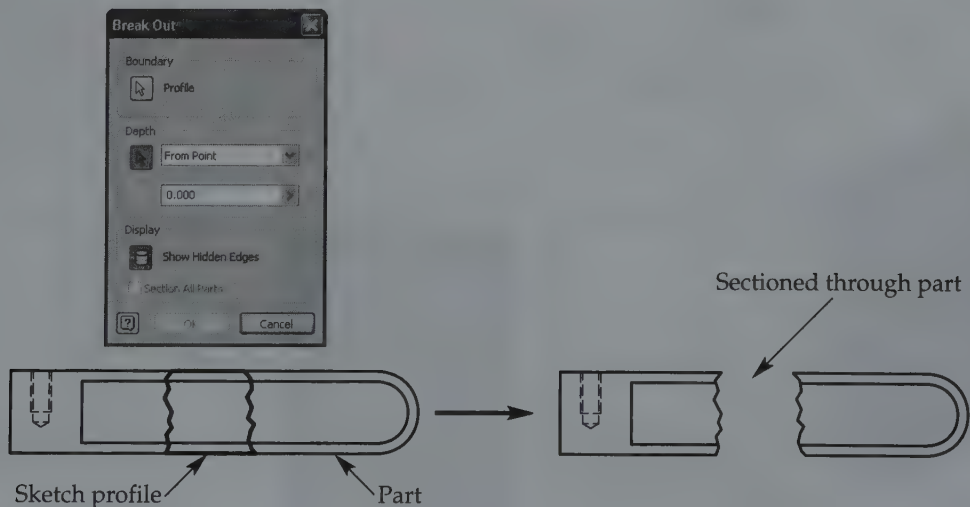
Figure 22-26.
Examples of acceptable revolved sections. Both examples require a full section created using the **Section View** tool, but only one example uses a broken-out section.

SECTION A-A   SECTION A-A
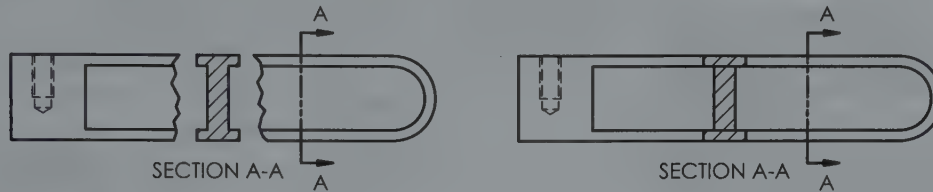
the broken-out section. Sectioning through a part is appropriate not only for assembly drawings, but also for part drawing applications, such as the *revolved section* shown in **Figure 22-26**.

> **revolved section:** A section view created by revolving it 90° in place, into a plane perpendicular to the line of sight.

Exercise 22-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-11.

Detail Views

> **Ribbon**
> **Place Views**
> **> Create**
>
> **Detail**
>
> **Shortcut**
> **Create View**
> **> Detail View...**

Use the **Detail View** tool to create a *detail view*, also known as a *view enlargement*. Create a detail view from an existing view to overcome the problem of displaying geometry that is small in relation to the overall size of the part. Select a view from which to extract the detail view, if you have not already, to display the **Detail View** dialog box. See **Figure 22-27**. Specify the view identifier and scale using the corresponding text boxes. The scale is twice the scale of the parent view by default. Use the **Toggle Label Visibility** button to display the view label, and use the **Edit View Label** button to make changes to the default label. Specify the style of view geometry in the **Style** area.

> **detail view (view enlargement):** A view that increases the scale of specific features to create a cleaner, easier-to-understand drawing and to improve dimension clarity.

Next, create a boundary, or fence, to identify the area taken from the parent view to create the detail. Pick the **Circular** button to create a circular fence or the **Rectangular** button to create a rectangular fence. Then apply the techniques shown in **Figure 22-27** to create the appropriate fence. Select the **Jagged** button to display standard short break lines with the detail, as shown in **Figure 22-27**, or pick the **Smooth** button if short break lines are not suitable for your application. When you select the **Smooth** button, you have the option of checking **Display Full Detail Boundary** and **Display Connection Line**, as shown in **Figure 22-28**. Pick a location in the graphics window to position the detail, or select the **OK** button.

PROFESSIONAL TIP

Edit a detail view and view label as you do other views. Occasionally, as shown in **Figure 22-29**, the boundary and boundary label display is not acceptable. Change the boundary size and location by dragging the green grips. Drag the boundary label to adjust the position. Adjusting the boundary automatically updates the detail view.
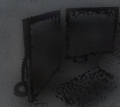
Figure 22-27.
Using the **Detail View** tool to create circular and rectangular detail view boundaries. The **Circular** option is the recommended ASME standard.

1. Select the center for the circle
2. Pick to define the circle radius and label position
3. Pick to position the detail

Circular Fence Shape

1. Select the center for the rectangle
2. Pick to define rectangle size
3. Pick to position the detail
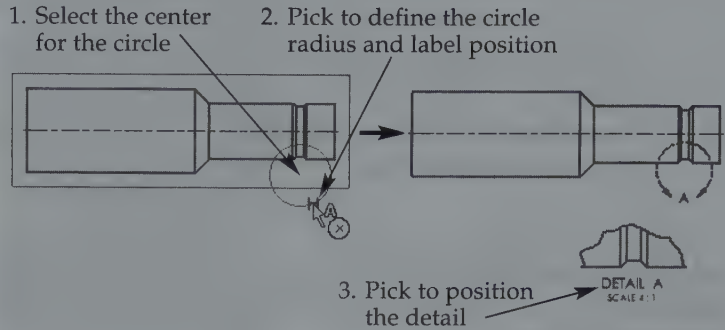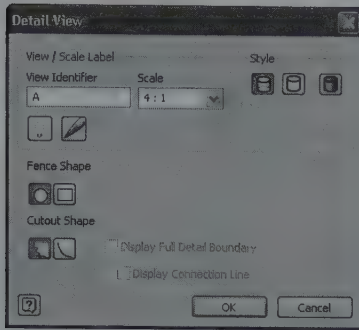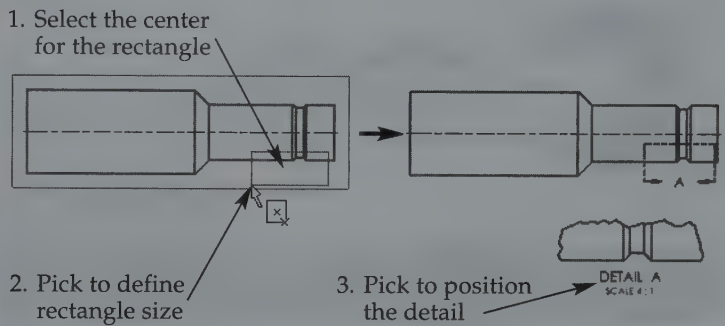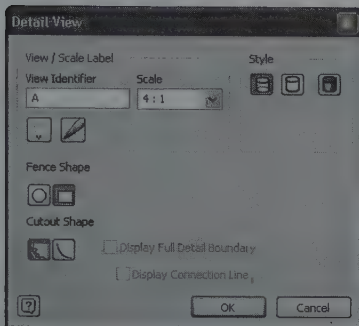
Rectangular Fence Shape

Figure 22-28.
Examples of options for using the **Smooth** cutout shape option, typically for non-ASME drafting applications.

DETAIL A
SCALE 4 : 1

☐ Display Full Detail Boundary
☐ Display Connection Line

DETAIL A
SCALE 4 : 1

☑ Display Full Detail Boundary
☐ Display Connection Line

DETAIL A
SCALE 4 : 1

☑ Display Full Detail Boundary
☑ Display Connection Line
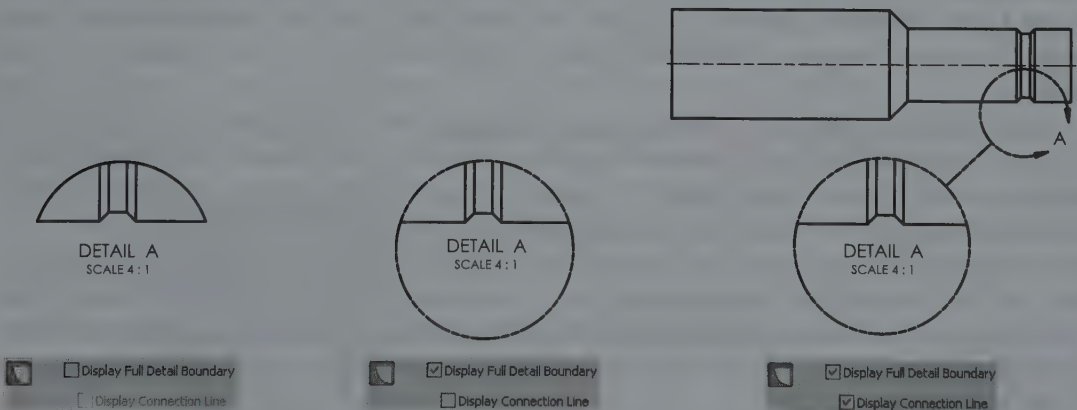
Figure 22-29.
Drag boundary grips and the label to a suitable size and position.

Click and drag to move

Click and drag to move label

Click and drag to resize

Label icon
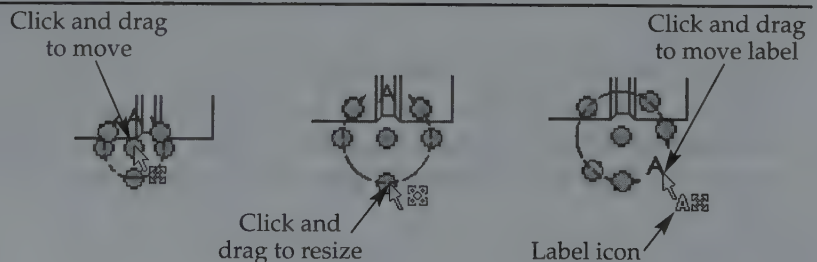
> **NOTE**
>
> Inventor includes a **Crop** tool that allows you to construct a rectangle on a view that crops, or hides, all objects outside of the rectangle. A unique linetype distinguishes cropped objects from visible object lines.
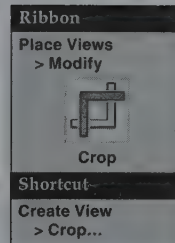
Exercise 22-12

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-12.

Conventional Breaks

Accessing the **Break** tool allows you to create a *conventional break*, often required when you are drawing a long component or to increase view scale without increasing sheet size. See **Figure 22-30**. Select a view, if you have not already, to display the **Broken View** dialog box. See **Figure 22-31A**. Pick the desired break style button from the **Style** area, and then adjust the orientation if necessary. Often Inventor chooses the correct orientation according to the selected view. If not, pick the **Horizontal Orientation** button to break a horizontal view, or pick the **Vertical Orientation** button to break a vertical view.

Use the **Min./Max.** *slider* to adjust the size and number of break edges for a rectangular-style break, or the break symbol size for a structural-style break. The **Gap** text box allows you to specify the distance between break lines. The **Symbols** text box, available when you specify the structural style, allows you to adjust the number of break symbols along the break lines. **Figure 22-31** shows examples of breaking the same view using different options. Then define the *break* by picking the location of the first break line and second break lines, as shown in **Figure 22-31A**. The break forms between the first and second lines when you pick the second line.

To edit the break, right-click on the break lines and select **Edit Break....** To change the location of the break without adjusting the length of the view, drag the green grip that appears when you hover over the break lines, or select the break lines. Shorten the break by dragging one of the break lines away from the center of the break. Extend the break by dragging one of the break lines toward and past the center of the break.

conventional break (break): The process of shortening the view of a long part that has a constant shape.

slider: A movable bar that increases or decreases a value when you slide the bar.

break: The portion of a view that you remove when you create a conventional break.

> **NOTE**
>
> Inventor references the same layer for both break styles, even though the **Rectangular Style** (short break lines) should have thick lines and **Structural Style** (long break lines) should have thin lines. Assigning a thin line layer to all break lines is more acceptable than using a layer with thick lines. Inventor does not provide options for creating the ASME-preferred appearance for cylindrical and tubular breaks. Breaking a view does not change the parametric size or shape of the model or drawing.
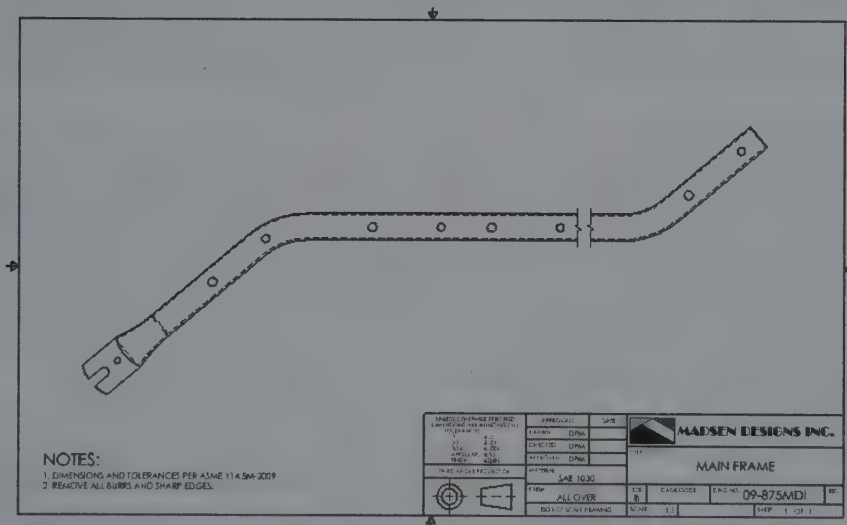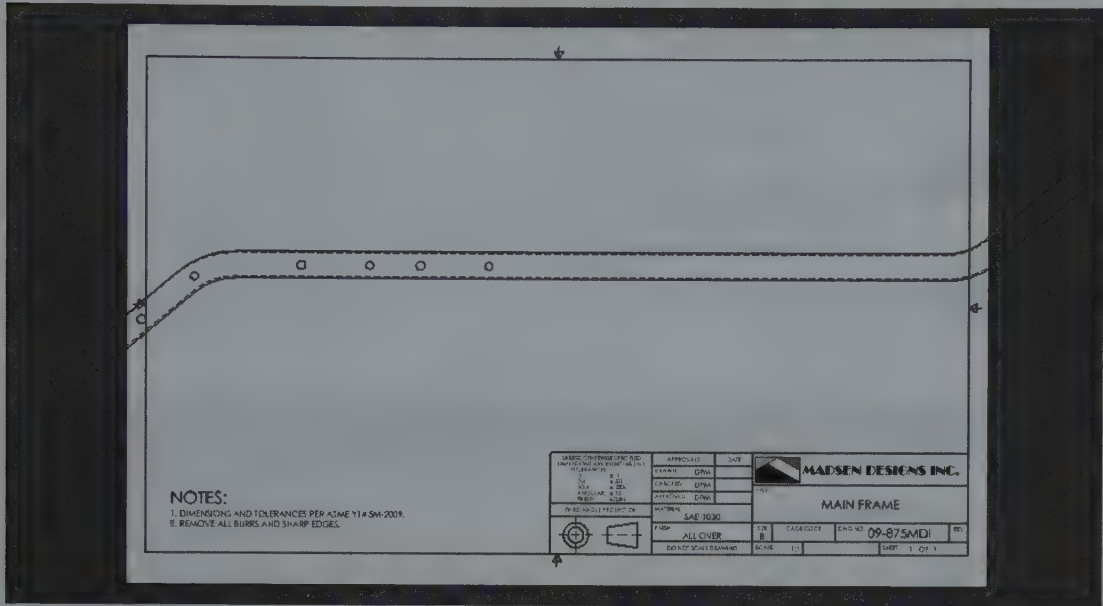
Figure 22-31.

A—Using the **Break** tool to create a conventional break using the structural style, horizontal orientation, average scale, small gap, and one symbol. B—Rectangular style, horizontal orientation, minimum scale, and small gap. C—Structural style, horizontal orientation, minimum scale, small gap, and two symbols. D—Rectangular style, vertical orientation, maximum scale, and small gap. E—Structural style, vertical orientation, maximum scale, large gap, and one symbol.
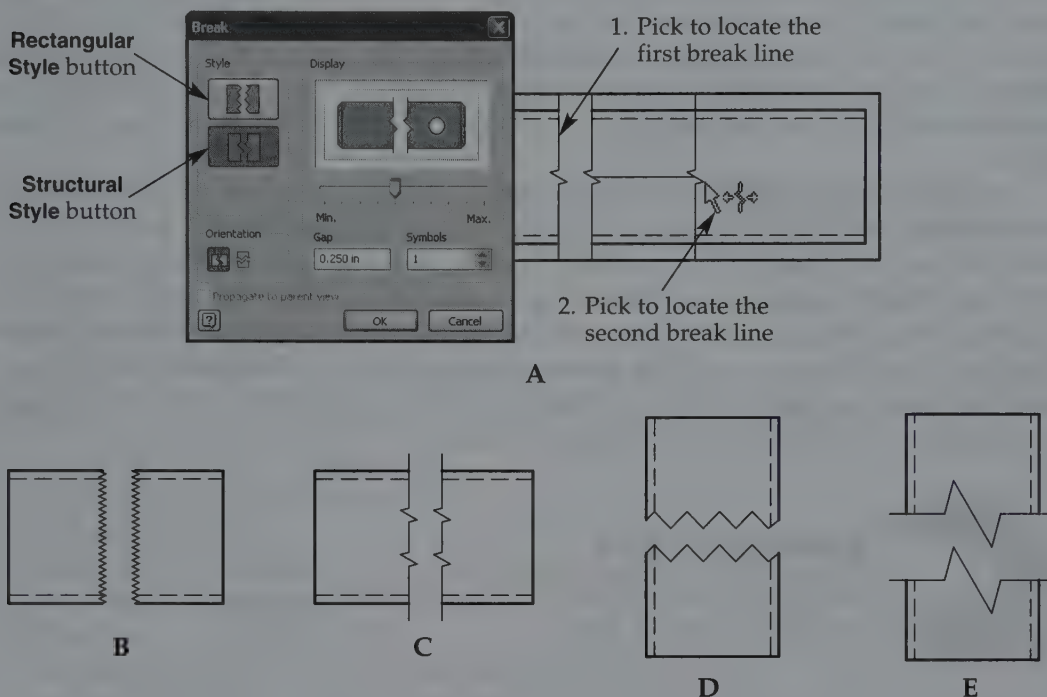
Exercise 22-13

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-13.

Sketched Views

The primary purpose of an Inventor drawing is to create 2D drawings that reference 3D models. However, if desired, you can use the **Draft View** tool to produce drawing views using sketch tools. The **Draft View** dialog box allows you to specify the view identifier and scale using the corresponding text boxes. Use the **Toggle Label Visibility** button to display the view label, and use the **Edit View Label** button to make changes to the default label. Pick the **OK** button to enter the sketch environment. Use sketch tools to create objects as you do in the model and other drawing sketch environments.

Ribbon
Place View > Create

Draft

NOTE

When you finish a draft view and exit the sketch environment, dimensional constraints are converted to drawing dimensions.

sheet format:
Multi-element sheet template stored in the drawing file.

A *sheet format* allows you to create a new sheet with common sheet items such as border and title block, but also with predefined views. To create a sheet format, develop a sheet with all the elements you want to reproduce when you use the sheet format, including a common arrangement of views. The views you insert are not displayed when you create a new sheet and act only as a view reference. Then right-click on the sheet in the browser and select **Create Sheet Format...** to display the **Create Sheet Format** dialog box. See **Figure 22-32A**. Enter a descriptive name, such as B-SIZE Front, Top, Right-side in the **Format Name** text box, and pick the **OK** button to create the sheet format.

The **Sheet Formats** folder of the **Drawing Resources** folder in the browser stores sheet formats, which you usually include in a template. See **Figure 22-32B**. To use a sheet format, double-click on the sheet format, or right-click the sheet format and select **New Sheet** to display the **Select Component** dialog box. See **Figure 22-32C**. If you previously inserted a component, the model is available in the **Document Name** drop-down list. Otherwise, pick the **Browse** button to display the **Open** dialog box and locate the model to insert. Then pick the **OK** button to create the drawing complete with a border, title block, and the specified views.
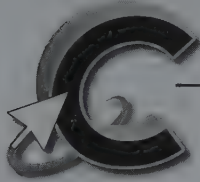
Exercise 22-14

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 22-14.

Figure 22-32.
A— Enter the name of the new sheet in the **Format Name** text box of the **Create Sheet Format** dialog box.
B— The **Sheet Formats** folder contains the templates used in drawings.
C— Right-click on a sheet format and select **New Sheet** to open the **Select Component** dialog box.
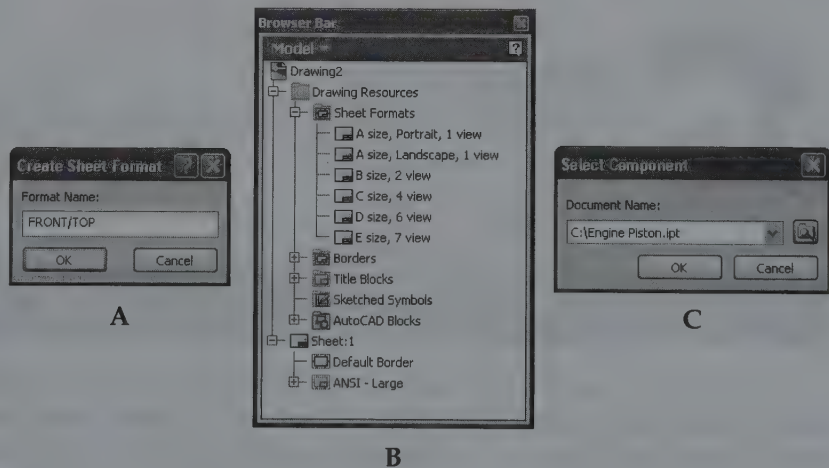
Chapter Test

1. What are detail drawings?
2. Define *monodetail*.
3. Define *multidetail*.
4. What does a drawing sheet represent?
5. What does the active drawing standard control?
6. Which ASME standard controls the appearance of broken-out sections?
7. Describe general notes and identify where they are located on a drawing.
8. What is a base view?
9. Describe a multiview drawing.
10. List the characteristics of the front view.
11. Describe an auxiliary view.
12. What is a partial auxiliary view?
13. Define *section view* and list two alternate names for a section view.
14. Briefly describe a cutting plane.
15. What is a cutting-plane line?
16. Define *full section*.
17. Describe a half section.
18. What is an offset section?
19. Describe an aligned section.
20. Explain the appearance and function of section lines.
21. Discuss the function of a broken-out section view.
22. What is a revolved section?
23. What is a detail view?
24. What is the purpose of a conventional break?
25. Briefly describe a sheet format.

Problems

Instructions:

- *Use the specified drawing template to develop the drawings shown from existing or new models. Select the correct view orientation when placing base views, which may not correspond to the orientation name in the **Base View** dialog box.*
- *Include a border and title block appropriate for the sheet size, available in the specified template file. Add all prompted entries as shown. Adjust the general notes as needed.*
- *If you notice problems with drawing views due to model issues or lack of model properties, open the model to make changes.*
- *Follow the specific instructions for each problem to create the drawing views. Do not add dimensions or annotations.*

1. **Title:** SWIVEL BOLT
   Units: Inch
   Template: Drawing-IN.idw
   View File: P9-2.ipt
   Save as: P22-1.idw
   Specific Instructions: Insert the front view as the base view and then project the isometric view shown. Edit the isometric view to display a shaded style.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES (IN.) TOLERANCES:	APPROVALS	DATE
.X ± .1	DRAWN DPM	
.XX ± .01	CHECKED DAM	
.XXX ± .005	APPROVED DAM	
ANGULAR: ± 1°		
FINISH: 62μIN.		

MADSEN DESIGNS INC.

TITLE SWIVEL BOLT

THIRD ANGLE PROJECTION

MATERIAL SAE 4140

FINISH ALL OVER

DO NOT SCALE DRAWING

SIZE A

CAGE CODE

DWG NO. IAA-009-02

REV 0

SCALE 2:1

SHEET 1 OF 1

2. **Title:** SLIDER HINGE CONNECTOR
   Units: Inch
   Template: Drawing-IN.idw
   View File: P10-6.ipt
   Save as: P22-2.idw
   Specific Instructions: Insert the left-side view as the base view and then project the front, top, and isometric views shown.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
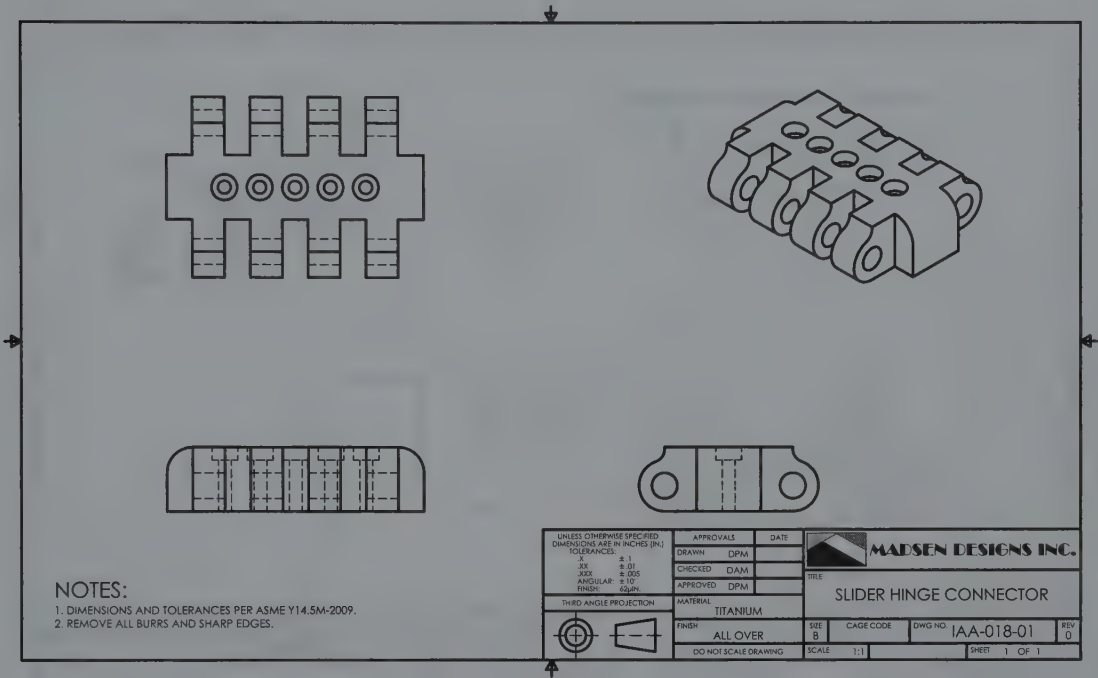
3. **Title:** BRACKET
   Units: Metric
   Template: Drawing-mm.idw
   View File: P7-7.ipt
   Save as: P22-3.idw
   Specific Instructions: Insert the front view as the base view and then project the right-side, top, and isometric views shown.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
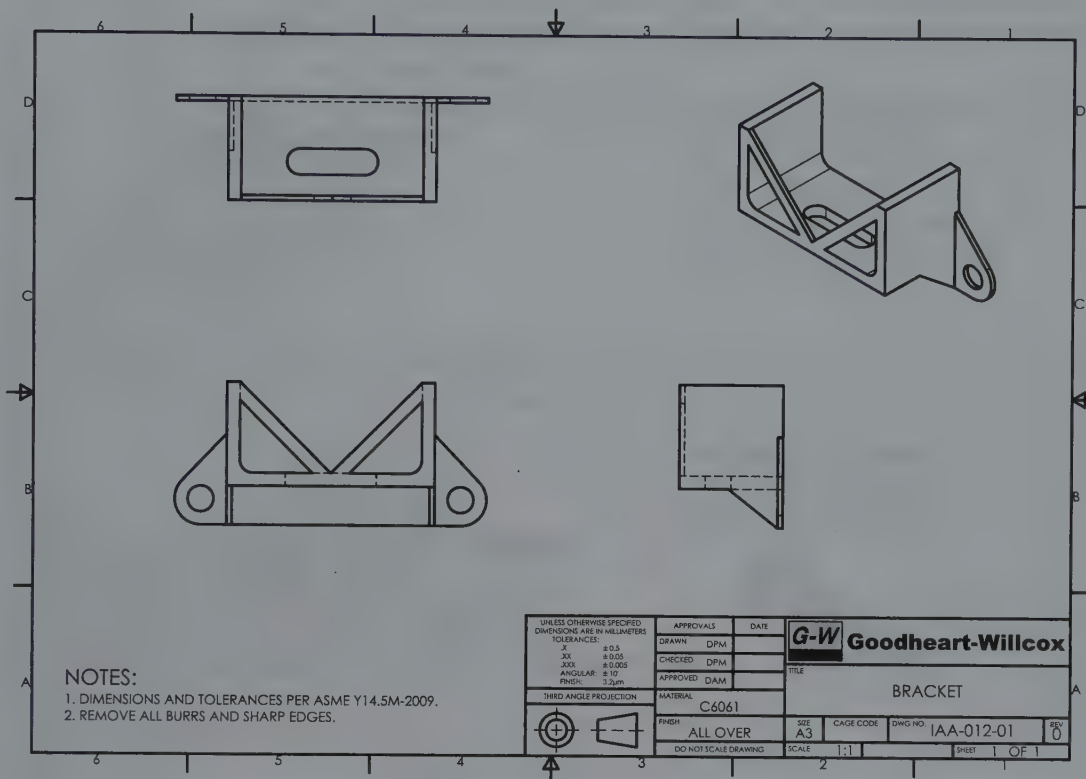
4. **Title:** C-CLAMP SWIVEL
   Units: Metric
   Template: Drawing-mm.idw
   View File: P8-2.ipt
   Save as: P22-4.idw
   Specific Instructions: Insert the top view as the base view and then create a full section as shown.

SECTION A-A

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN MILLIMETERS TOLERANCES:	APPROVALS		DATE	G-W Goodheart-Willcox		
.X ±0.5	DRAWN	DPM				
.XX ±0.05	CHECKED	DPM		TITLE		
.XXX ±0.005	APPROVED	DPM		C-CLAMP SWIVEL		
ANGULAR: ±10' FINISH: 3.2µm						
THIRD ANGLE PROJECTION	MATERIAL SAE 1035					
	FINISH ALL OVER			SIZE A3	CAGE CODE	DWG NO. IAA-001-02	REV 0
	DO NOT SCALE DRAWING			SCALE 2:1		SHEET 1 OF 1
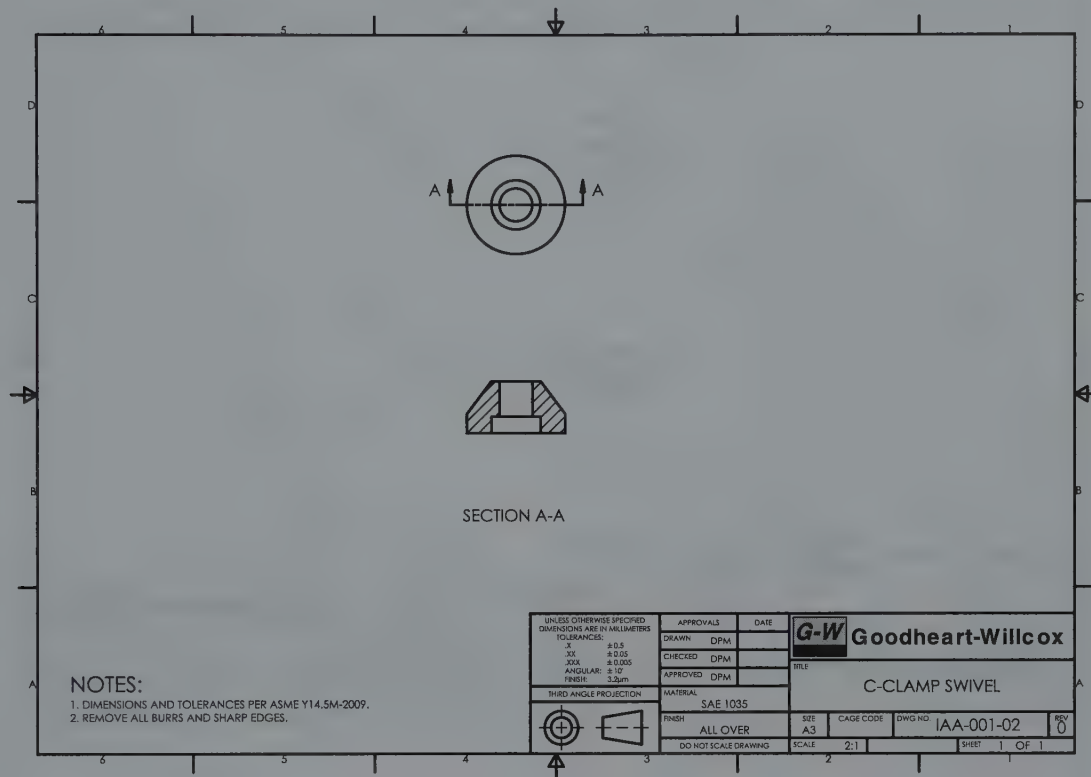
5. **Title:** C-CLAMP SWIVEL
   Units: Metric
   Template: Drawing-mm.idw
   Specific Instructions: Open file P22-4.idw and save it as P22-5.idw. In the P22-5.idw file, edit the cutting-plane line sketch, probably Sketch1 in the browser, to create the half section shown. Turn off the visibility of the vertical lines as shown.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
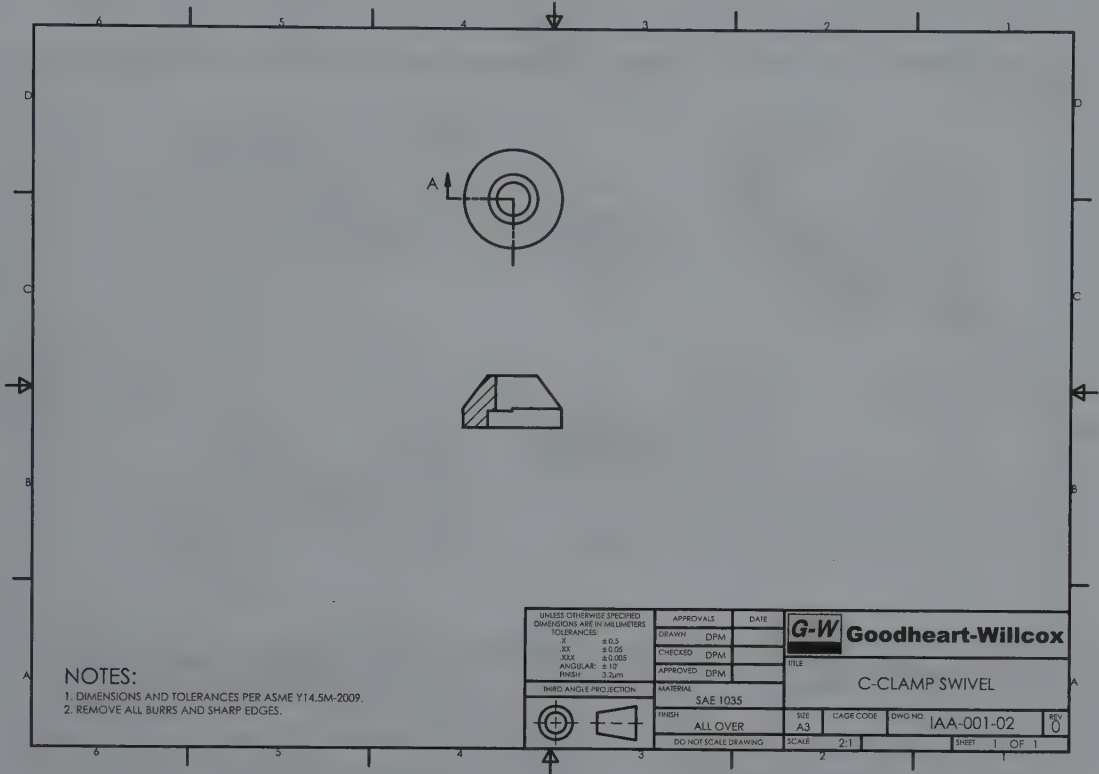
6. **Title:** BENT PULLER
   Units: Inch
   Template: Drawing-IN.idw
   View File: P7-5.ipt
   Save as: P22-6.idw
   Specific Instructions: Insert the front view as the base view and then project the top and isometric views shown. Create the auxiliary view shown.

Problems – Chapter 22
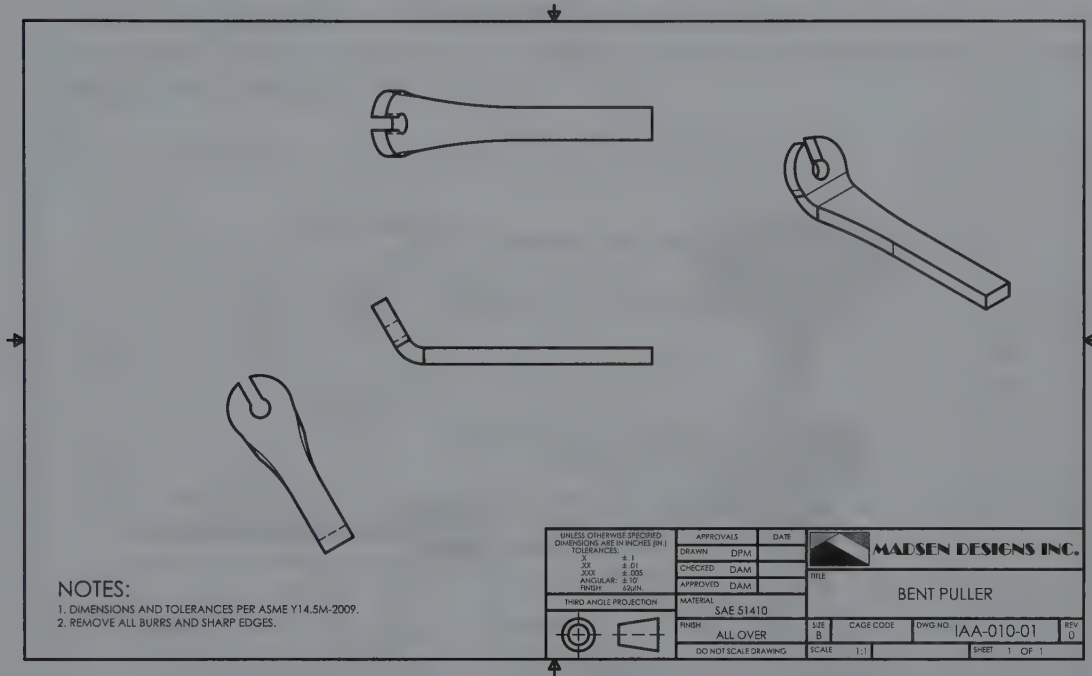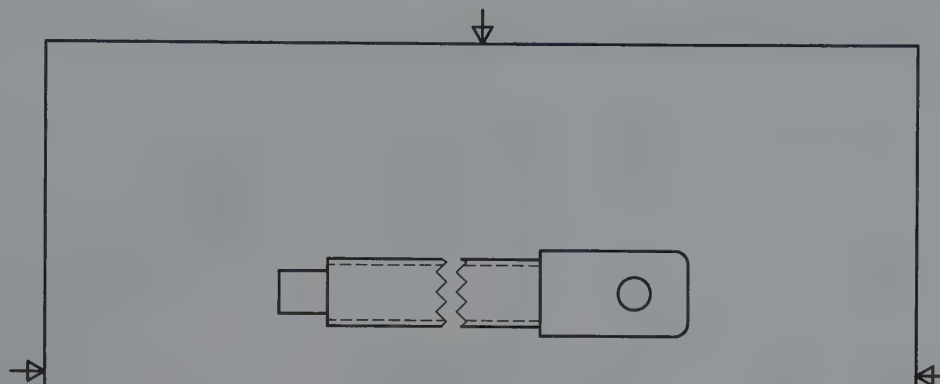
7. **Title:** SCREW
   Units: Metric
   Template: Drawing-mm.idw
   View File: P11-1.ipt
   Save as: P22-7.idw
   Specific Instructions: Insert the front view as the base view and then create the conventional break shown.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN MILLIMETERS	APPROVALS		DATE	**G-W Goodheart-Willcox**				
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	CHECKED	DPM		TITLE				
	APPROVED	DPM		SCREW				
THIRD ANGLE PROJECTION	MATERIAL SAE 1035							
	FINISH ALL OVER			SIZE A	CAGE CODE	DWG NO. IAA-001-02		REV 0
	DO NOT SCALE DRAWING			SCALE 2:1		SHEET 1 OF 1		

8. **Title:** FUNNEL
   Units: Inch
   Template: Drawing-IN.idw
   View File: P9-3.ipt
   Save as: P22-8.idw
   Specific Instructions: Insert the top view as the base view and then create a full section as shown. Project the shaded isometric view from the section as shown. Turn off the visibility of edges and edit the view to include the label. Create a base "front" view from which to project the shaded isometric view shown. Delete the base "front" view and turn off the visibility of edges as shown. Edit the general notes to add the third note.

A ◄───────►A

SECTION A-A

SECTION A-A

NOTES:

1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
3. ALL FILLETS AND ROUNDS R.031.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES (IN.) TOLERANCES		APPROVALS	DATE	MADSEN DESIGNS INC.
X ± .1		DRAWN DPM		
XX ± .01		CHECKED DAM		TITLE
XXX ± .005		APPROVED DAM		FUNNEL
ANGULAR ± 10'				
FINISH 63 μIN.				
THIRD ANGLE PROJECTION	MATERIAL BLACK ACETAL RESIN			
	FINISH ALL OVER			SIZE B	CAGE CODE	DWG NO. IAA-008-01	REV 0
	DO NOT SCALE DRAWING			SCALE 1:1		SHEET 1 OF 1

9. **Title:** 45 DEGREE SPLIT ELBOW
   Units: Metric
   Template: Drawing-mm.idw
   View File: P11-3.ipt
   Save as: P22-9.idw
   Specific Instructions: Insert the front view as the base view and then project the left-side, bottom, and isometric views shown. Create the removed auxiliary VIEW A-A and the removed VIEW B-B using the **Auxiliary View** tool. Add the full section shown, and edit views as needed.
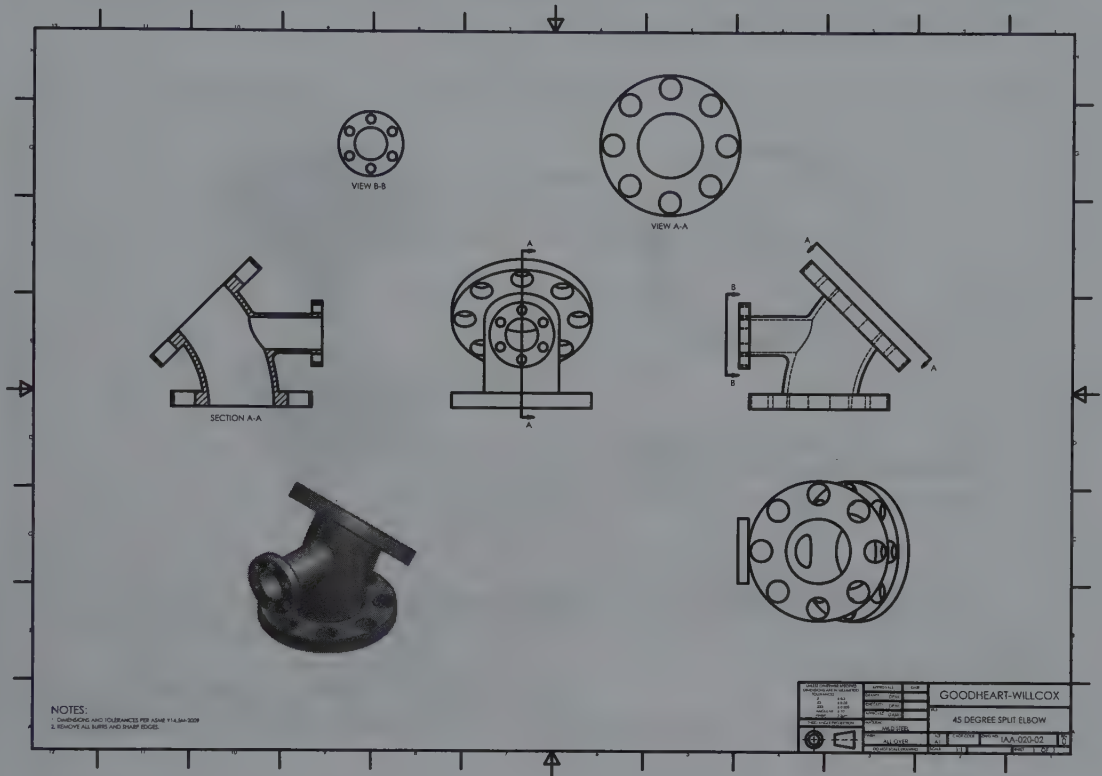
10. **Title:** CLIP
    Units: Inch
    Template: Drawing-IN.idw
    View File: P15-5.ipt
    Save as: P22-10.idw
    Specific Instructions: Use the **Base View** tool to create the flat pattern view shown. If you did not create a flat pattern model, you must do so before you can create a flat pattern drawing view. Access the **Base View** tool again and place the front view as shown. Project the isometric view to complete the drawing views.

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES (IN.)
TOLERANCES:
.X   ± .1
.XX   ± .01
.XXX   ± .005
ANGULAR: ± 1°
FINISH: 63μIN.
THIRD ANGLE PROJECTION

APPROVALS		DATE
DRAWN	DPM	
CHECKED	DAM	
APPROVED	DAM	

MADSEN DESIGNS INC.

TITLE
CLIP

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11 GAGE 1010
FINISH
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| SIZE B | CAGE CODE - | DWG NO. 1AA-036-01 | REV 0 |
| SCALE 1:1 | | SHEET 1 OF 1 | |

DO NOT SCALE DRAWING

11. **Title:** HANDLE
    Units: Inch
    Template: Drawing-IN.idw
    View File: P22-11.ipt
    Save as: P22-11.idw
    Specific Instructions: Use the drawing shown to create a part model. Save the part as P22-11.ipt and then use the model to create the drawing views shown. Do not add the centerlines or dimensions.

Ø.201 ▼ .625
1/4-20 UNC - 2B ▼ .500

.500
.250
.375

4.500
.875
.250
.125
A
R.500
R.375

SECTION A-A
A

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES (IN.) TOLERANCES:		APPROVALS		DATE	*G-W* **Goodheart-Willcox**		
X ± .1		DRAWN	DPM				
XX ± .01		CHECKED	DAM		TITLE		
XXX ± .005		APPROVED	DAM			HANDLE	
ANGULAR ± .10							
FINISH: 63μin.							
THIRD ANGLE PROJECTION		MATERIAL	SAE 1030				
		FINISH			SIZE B	CAGE CODE	DWG NO P18-10	REV
		ALL OVER					
		DO NOT SCALE DRAWING			SCALE 1:1		SHEET 1 OF 1

CHAPTER 23
Dimensioning Part Drawings

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Apply model and drawing dimensions to part drawings.
✓ Add centerlines and center marks.
✓ Use the **General Dimension** tool.
✓ Edit and arrange dimensions.
✓ Use the **Baseline** and **Ordinate Dimension** tools.
✓ Add notes, symbols, and tables.

This chapter explains how to use drawing annotation tools and options to add *dimensions*, notes, and text information to drawing views. You will explore options for extracting model parameters and creating drawing dimensions. The focus of this chapter is dimensioning part drawings, but you will apply many of the same tools and options to develop assembly and weldment drawings.

dimensions: Text and graphics that define the size and shape of object features. Along with notes, dimensions specify the location and characteristics of geometry and surface texture to drawing views.

Dimension Fundamentals

The model that you reference to create drawing views includes parameters that you can extract and use as *model dimensions*. See **Figure 23-1**. You can edit parametric model dimensions in the drawing to make changes to the size and shape of the model. *Drawing dimensions* allow you to dimension a drawing when model dimensions are not appropriate, or when additional dimensions are required.

model dimensions: Model parameters, such as dimensional constraints and feature specifications, that are available to use as dimensions on a drawing.

NOTE

This chapter focuses on dimensioning orthographic views, but you can apply the same tools and options to dimension isometric views.

drawing dimensions: Dimensions you place as an alternative or in addition to model dimensions to fully describe the component.

Figure 23-1.
An example of model dimensions extracted to a drawing view.

Model Dimensions (Dimensional Constraints)   Extracted Model Dimensions
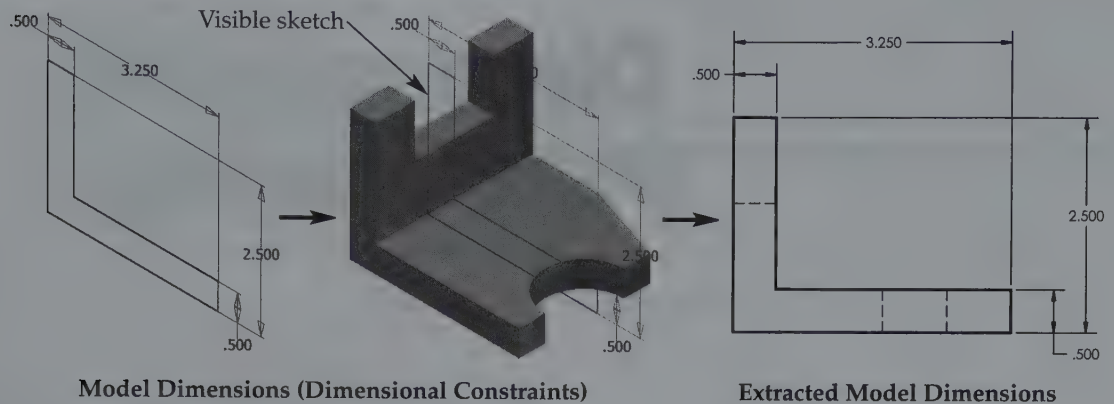
Model Dimensions

You can only extract, or retrieve, model dimensions that are planar to the drawing view. For example, you can show the model dimension that defines the part thickness in the view that displays the thickness. You can add model dimensions during view placement by selecting the **Retrieve all model dimensions on view placement** check box on the **Drawing Tab** of the **Options** dialog box before you create a base view. The **All Model Dimensions** check box on the **Display Options** tab of the **Drawing View** dialog box also allows you to display model dimensions when you insert or edit a base view.

Automatic means of retrieving model dimensions are effective for some applications. However, often you need additional control over the dimensions that appear, or want to acquire model dimensions that were formed after initial view placement. Access the **Retrieve Dimensions** tool to display the **Retrieve Dimensions** dialog box, shown in **Figure 23-2**. Use the **Select View** button to select a view referencing a model. To retrieve dimensions for specific model features, pick the **Select Features** radio button and then choose the features. To retrieve all part dimensions, pick the **Select Parts** radio button, and select the parts.

Selecting features allows you to filter out unneeded dimensions that appear when you select a part, while selecting a part displays all available model dimensions. Depending on the model, some dimensions may not be necessary or appropriate for the view. Pick the **Select Dimensions** button to choose only the model dimensions to display in the drawing. Pick the **Apply** button to create dimensions and remain in the tool, or pick the **OK** button to create the dimensions and exit.

Ribbon
Annotate
> Dimension

Retrieve

Shortcut
Retrieve
Dimensions...
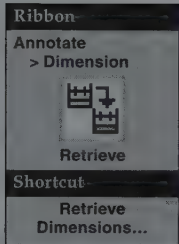
Exercise 23-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-1.

Figure 23-2.
Use the **Retrieve Dimensions** dialog box to retrieve model dimensions associated with a specific view.
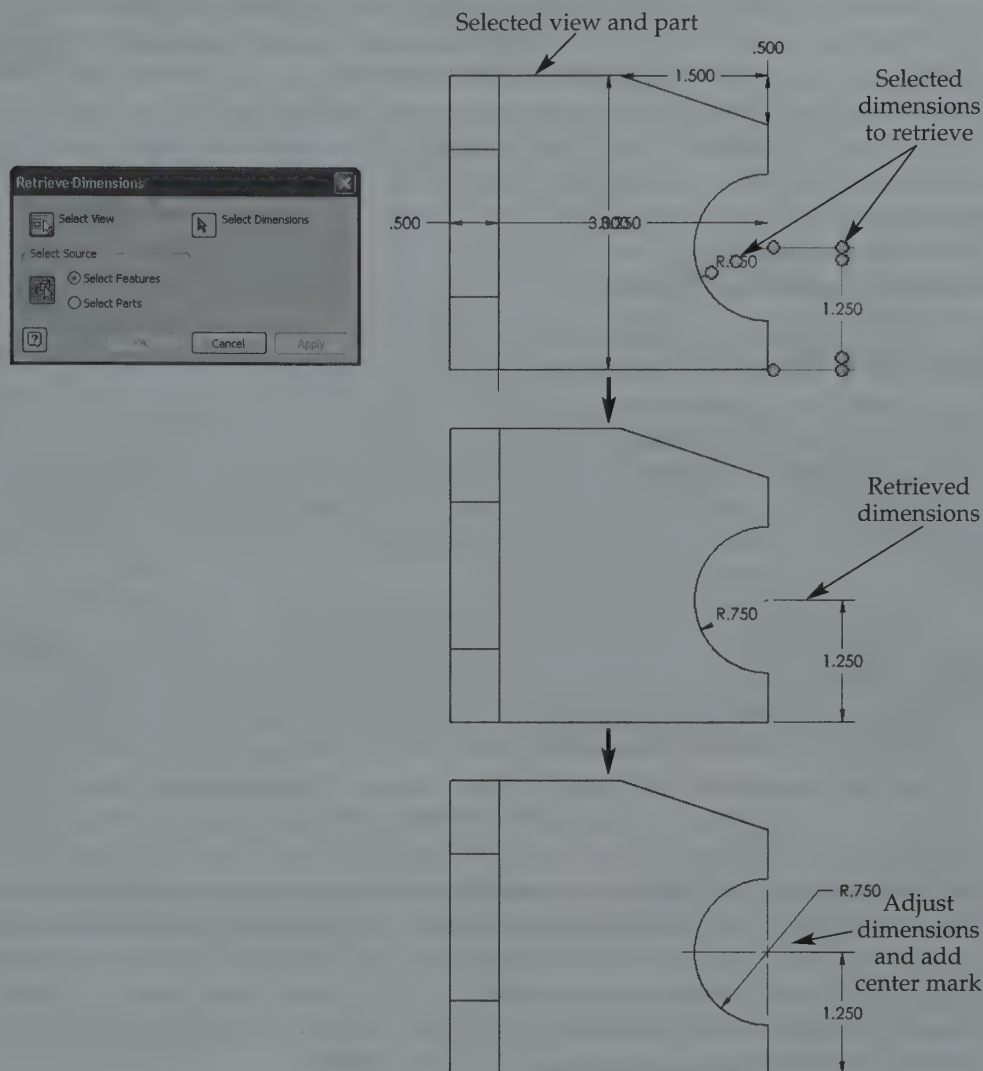
Drawing Dimensions

Add drawing dimensions when model dimensions do not fully document design intent or are not appropriate according to correct drafting practices. Inventor provides many tools and options for annotating a drawing. Tools are available for specific dimensioning methods, and you can add the dimensions, notes, symbols, and tables required for most drawing applications. You cannot control model parameters using drawing dimensions. However, drawing dimensions are associated with model geometry. If you modify model parameters in the model file drawing and in the model, the dimensions in the drawing adjust to the changes.

> **NOTE**
>
> Settings assigned to the active standard and annotation tools determine the appearance of annotations. However, you are responsible for adjusting the display and, in some cases, overriding standard defaults to create a drawing that is clear, readable, and prepared according to appropriate drafting standards.

Drawings that contain circular geometry or symmetrical features require center marks and centerlines in order to dimension and describe the center and location of features. You can use the **Automated Centerlines** tool to add center marks and centerlines to specified objects. Other center mark and centerline tools are also available that you can use in addition to or as an alternative to automated centerlines. It is usually easiest to add center marks and centerlines before adding dimensions and other annotations.

Automated Centerlines

Right-click on a view and select **Automated Centerlines...** to access the **Automated Centerlines** dialog box. See **Figure 23-3**. The dialog box allows you to specify which items in the view automatically receive centerlines and center marks. Adjusting values in the **Automated Centerlines** dialog box overrides the default settings specified using the **Automated Centerlines...** button on the **Drawing** tab of the **Document Settings** dialog box. If you set appropriate automated centerline document settings and save the settings in a template, you can pick the **OK** button immediately to add centerlines and center marks for most applications.

Pick buttons in the **Apply To** area corresponding to model features that will receive centerlines and center marks. In the **Projection** area, pick the **Objects in View, Axis Normal** button to add center marks to features normal to the view plane, and the **Objects in View, Axis Parallel** button to add centerlines to features parallel to the view plane. The **Radius Threshold** area allows you to apply size limits to features that will receive centerlines and center marks. Use the **Precision** drop-down list to set the threshold limit precision. Then use the **Fillets:** text boxes to specify the minimum and maximum size of fillet features that will receive centerlines and center marks. Use the **Circular Edges:** text boxes to specify the minimum and maximum size of other circular features that receive centerlines and center marks. For example, with precision set to 3.123, if you specify a minimum size of .500 and a maximum size of 0, only features that are greater than or equal to .500 receive centerlines and center marks. The **Arc Minimum:** text box allows you to specify a minimum size for arcs, circles, and ellipses that will receive centerlines and center marks.

Figure 23-3.
Adding centerlines and center marks to a view using the **Automated Centerlines** tool.
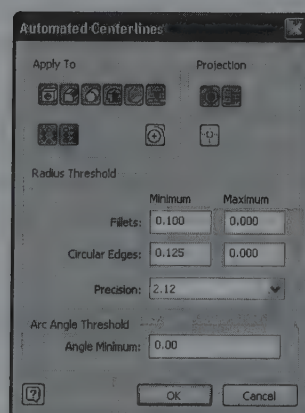
Selected view
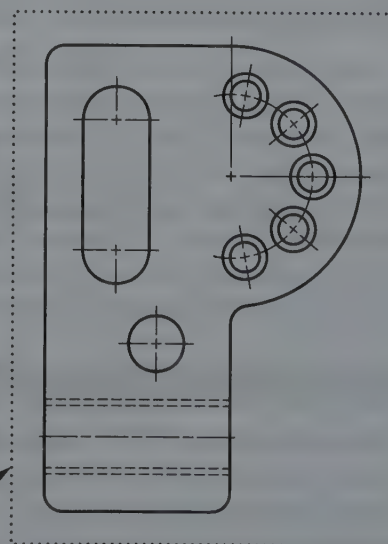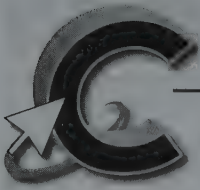
Exercise 23-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-2.

Centerline and Center Mark Tools

Use the **Center Mark** tool to add a center mark to an object representing a circular feature or a sheet metal punch feature. Select a circular edge or center point as shown in **Figure 23-4A**. Select the features that need center marks. Press [Esc] or right-click and select **Done** to exit. You can right-click before you select objects and deselect **Extension lines** to create a center mark without extension lines, or deselect **Punch Centers** to exclude selection of sheet metal punched features.

Access the **Centerline** tool to display a centerline between points, or to display center marks between circular features. See **Figure 23-4B**. To create a centerline, pick two edges or points, such as two grip points. To create connected center marks, pick two or more edges or center points. After you select objects, press [Enter] or right-click and pick **Create**. Press [Esc] or right-click and select **Done** to exit.

Use the **Centered Pattern** tool to identify the center of a circular pattern and add center marks to pattern occurrences. See **Figure 23-4C**. First, specify the center of the pattern by picking the pattern center point or the circular edge concentric to the pattern, if available. Then select the objects in the pattern in sequential order. After you select objects, right-click and pick **Create**. Press [Esc] or right-click and select **Done** to exit the tool.

Ribbon

Annotate
> Symbols

Center Mark

Ribbon

Annotate
> Symbols

Centerline

Ribbon

Annotate
> Symbols
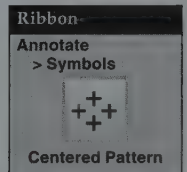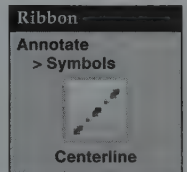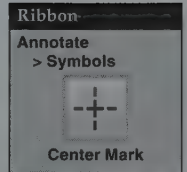
Centered Pattern

Figure 23-4.
Examples of manually created centerlines and center marks. A—Using the **Center Mark** tool. B—Using the **Centerline** tool. C—Using the **Centered Pattern** tool. D—Using the **Centerline Bisector** tool.
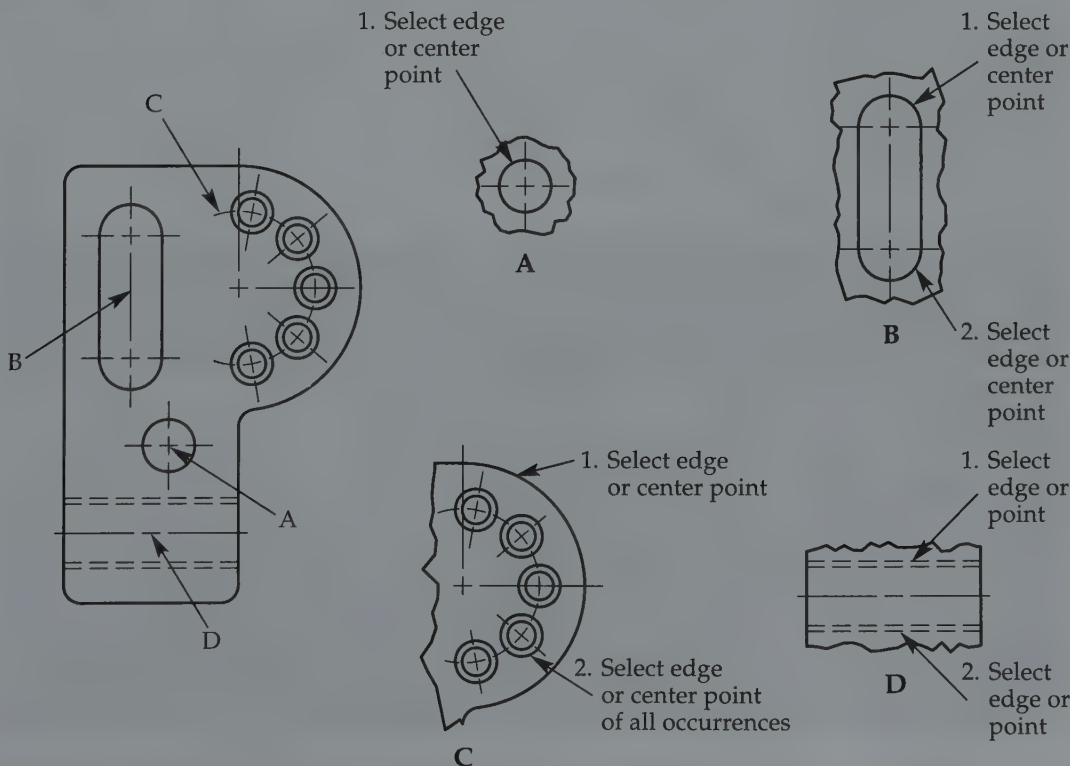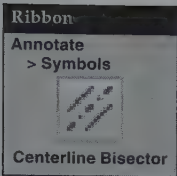
Access the **Centerline Bisector** tool to place a centerline through a feature parallel to the view plane. See **Figure 23-4D**. Select two edges, such as hidden or sectioned hole lines, and then press [Enter] or right-click and select **Create**. Press [Esc] or right-click and select **Done** to exit.

Editing Centerlines and Center Marks

You can make changes to centerlines and center marks by moving or stretching size and location grips or by applying shortcut menu options. **Figure 23-5A** shows the grips that appear when you select a centerline or center mark. Drag the end of a centerline or center mark to increase or decrease the length. You will notice that the end snaps to the correct extension as you near an edge, or becomes hidden when you near the center point. You can use the center point grip to move the center mark to another point, but leaving the center mark in space orphans the center mark. **Figure 23-5B** describes options that may be available when you right-click on a centerline or center mark.

> **PROFESSIONAL TIP**
>
> You can often avoid adding or removing dashes by using the appropriate tool, such as the **Centerline** tool.

Exercise 23-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-3.

Figure 23-5.
A—Examples of editing centerlines and center marks. B—Options that may be available when you right-click on a centerline or center mark.

Drag to another feature to redefine · Drag to another feature to redefine · Drag to extend or reduce · Reduce to center · Extend past extension · Drag to extend or reduce

A
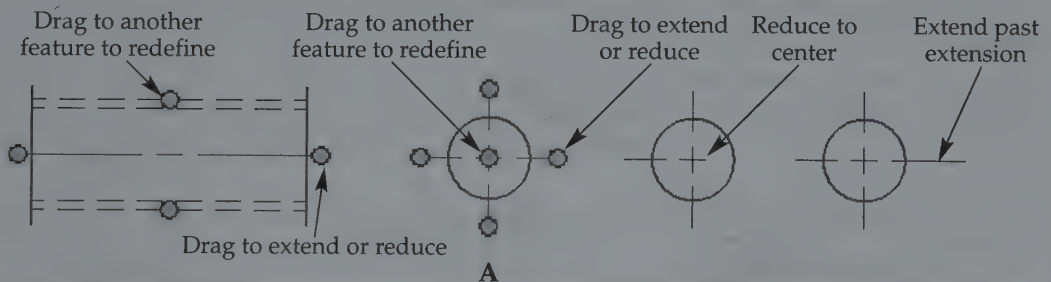
Option	Description
Delete [Delete]	Deletes the selection.
Visibility	Turns off the visibility of the selection.
Edit>Extension Lines	Deselect to hide extension lines.
Edit>Align to Edge	Rotates the center mark to align with the selected edge.
Edit>Fit Center Mark	Returns extension lines to the extents of the object.
Edit>Add/Remove Dashes	Adds or removes dashes along an extension line at the selected points; required for long extension lines.
Edit>Uniform Dashing	Applies uniform spacing between dashes on a selected extension line after dashes have been added or removed.

B

General Dimension Tool

The **General Dimension** tool allows you to create several types of drawing dimensions. See **Figure 23-6**. Dimension placement varies depending on the type of geometry you dimension. In general, select objects or points to dimension, and then pick a location for the dimension value. Select the actual surfaces from which the measurement originates when possible, or pick points. Be sure that you select the correct origin of the dimension so that the correct measurement and extension line offset appears. As you move the dimension value after selecting objects or points, the dimension appears dotted when you reach the standard offset. A dotted line or centerline appears as you near the center of the dimension line. Use these aids to help place the dimension. See **Figure 23-6**.
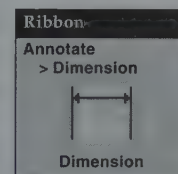
Figure 23-6.
Using the **General Dimension** tool to create linear, aligned, angular, diameter, and radius dimensions.

Dimension	Description	Origin Selection
Linear	Measures straight distances, such as dimensions between horizontal, vertical, or slanted surfaces.	One or two curves or points, or a curve and a point.
Aligned	Measures distances and places the dimension line parallel to a slanted surface.	One or two curves or points, or a curve and a point.
Angular	Measures the angle of a surface.	Two non-parallel lines.
Diameter	Measures the diameter of a circular feature, usually a circle.	Circular edge.
Radius	Measures the radius of a circular feature, usually an arc.	Circular edge.

General dimensions reference the active standard and form a dimension type based on the objects you select. If the dimension appearance does not appear correct, you may be able to adjust the dimension or apply a style override before selecting a location for the dimension value. Right-click to access dimension settings that vary depending on the type of dimension. Some settings are available from the **Dimension Type** or **Options** cascading submenus.

Edit Dimension Dialog Box

By default, the **Edit Dimension** dialog box appears when you pick a location for the dimension value. See **Figure 23-7**. Deselect the **Edit dimension when created** check box if you do not want to see the **Edit Dimension** dialog box when placing general dimensions. If the dimension value does not require modification, pick the **OK** button to create the dimension. Press [Esc], right-click and select **Done**, or access a different tool to exit.

For most applications, you should only use the options in the **Text** tab, shown in **Figure 23-7A**, to add a prefix or suffix to the associated value (<<>>), or to hide the value. For example, add a 4X in front of a value to dimension a repetitive feature, or ∅ in front of a linear diameter dimension. The **Precision and Tolerance** tab, shown in **Figure 23-7B**, includes options to adjust the tolerance precision and to apply a unique tolerance method for a specific dimension.

> **CAUTION**
>
> The **Precision and Tolerance** tab also allows you to override the associated value. Overriding the displayed dimension value disassociates the dimension value from the drawing view and model, which is usually inappropriate.
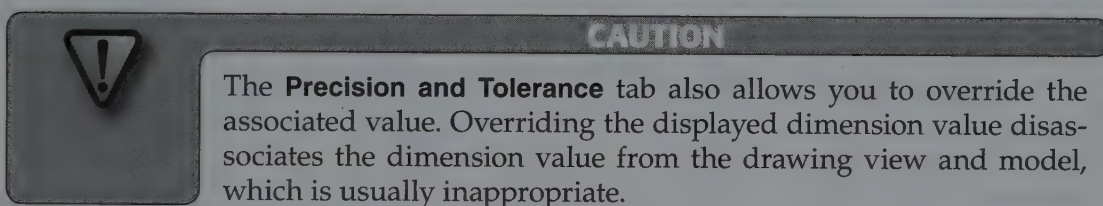
Inspection Dimensions

The **Inspection** tab of the **Edit Dimension** dialog box allows you to identify a dimension that requires testing throughout the design and manufacturing of a product. See **Figure 23-8**. Pick the **Inspection Dimension** check box to include test information.

Figure 23-7.
The **Edit Dimension** dialog box allows you to add content to or override the characteristics of a dimension value. A—Use the **Text** tab primarily to add a prefix or suffix or to hide the dimension value. B—Use the **Precision and Tolerance** tab to adjust the tolerance of a specific dimension.

Add content to the associated value

Apply a specific tolerance

Adjust the tolerance precision

A

B

Pick to add
a symbol

Then select the shape of the inspection dimension frame or choose not to include a frame by selecting the appropriate radio button. To include a label that identifies the dimension, type a label in the **Label** text box. Type an inspection rate in the **Inspection Rate** text box to indicate how often to perform a test on the dimension. The rate can have different meanings depending on the application. For example, 100% can mean to check the length of the part for tolerance every time the assembly receives the part.

Exercise 23-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-4.

Editing and Arranging Dimensions
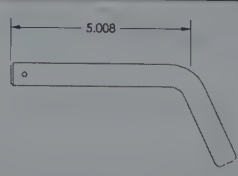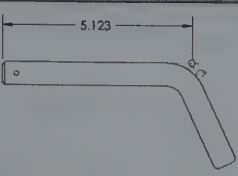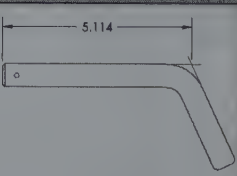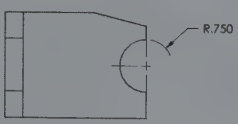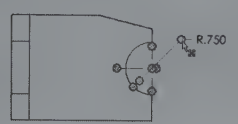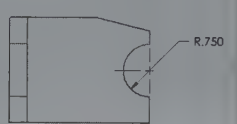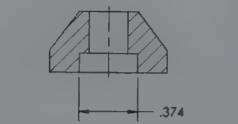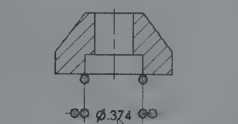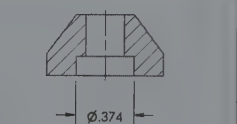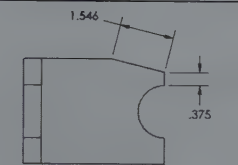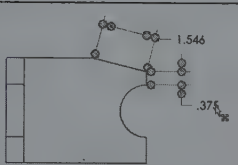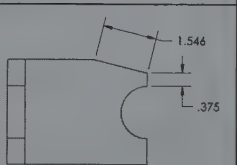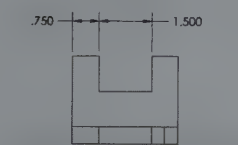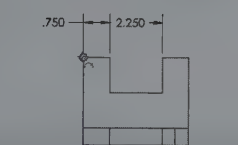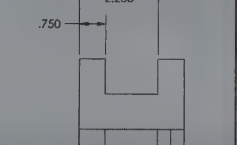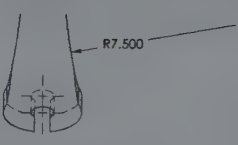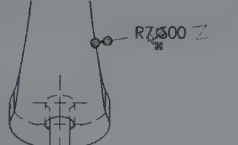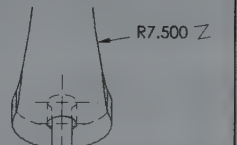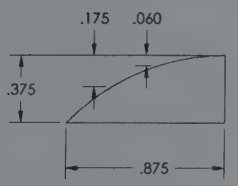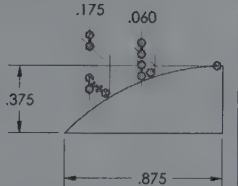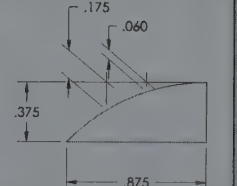
As you learn to dimension a drawing, you will find that the active standard and specific dimensioning tools usually create acceptable or near-acceptable dimensions. However, you will also find that many dimensions require modification to account for unique geometry in a view, or to meet drafting standards. Although you will learn additional dimensioning tools and options later in this chapter, you should become familiar with basic dimension editing and arrangement options before proceeding.

Often the key to making a dimension look the way you want is to experiment with options, such as moving the dimension value to a new location or selecting a shortcut menu option. You can modify dimensions by changing the placement and offset of value, adjusting display characteristics, or overriding dimension style variables. **Figure 23-9** shows a few of the many common dimension adjustments you may have to apply. When dragging a dimension value, use the dotted or center reference lines to pick an accurate location, and use point alignment between other dimensions or objects.

Right-click on a dimension to access several options for adjusting display characteristics, overriding default style variables, editing the dimension value, moving the dimension to a different view, and copying properties. Common applications include using the **Precision** cascading submenu to select a different tolerance precision and picking **Edit...** or **Text...** to add a prefix or suffix to the dimension value. Available settings vary depending on the selected dimension type. For example, the **Jogged** option is available when you right-click on a radius dimension used to dimension a large radius. Explore each option as needed to control the appearance of specific dimensions.

Original	Operation	Result	Description
5.008	5.123	5.114	Use point alignment while dragging the extension line endpoint to acquire the apparent intersection.
R.750	R.750	R.750	Stretch center mark extension lines and drag the dimension value to force the appearance shown.
.374	Ø.374	Ø.374	Deselect the **Arrowheads Inside** option and edit the value to add the Ø prefix.
1.546 / .375	1.546 / .375	1.546 / .375	Drag dimension values to reposition the value and add the shoulder.
.750 / 1.500	.750 / 2.250	2.250 / .750	Drag the extension line endpoint to a new point and drag the dimension value up.
R7.500	R7.500	R7.500	Select the **Jogged** option.
.175 .060 / .375 / .875	.175 .060 / .375 / .875	.175 .060 / .375 / .875	Drag the grip at the extension line/dimension line intersection to create oblique extension lines.

NOTE

Right-click on a view and select from the **Annotation Visibility** cascading submenu to toggle the visibility of model and drawing dimensions and other annotations.

Arrange Dimensions Tool

Use the **Arrange Dimensions** tool to align and offset dimensions according to the active standard spacing and dimension location. See **Figure 23-10.** You can arrange a group or combination of linear, angular, or coordinate dimensions. For the tool to work most effectively, dimensions should be in an approximate configuration. The

Figure 23-10.
Arranging dimensions using the **Arrange Dimensions** tool.
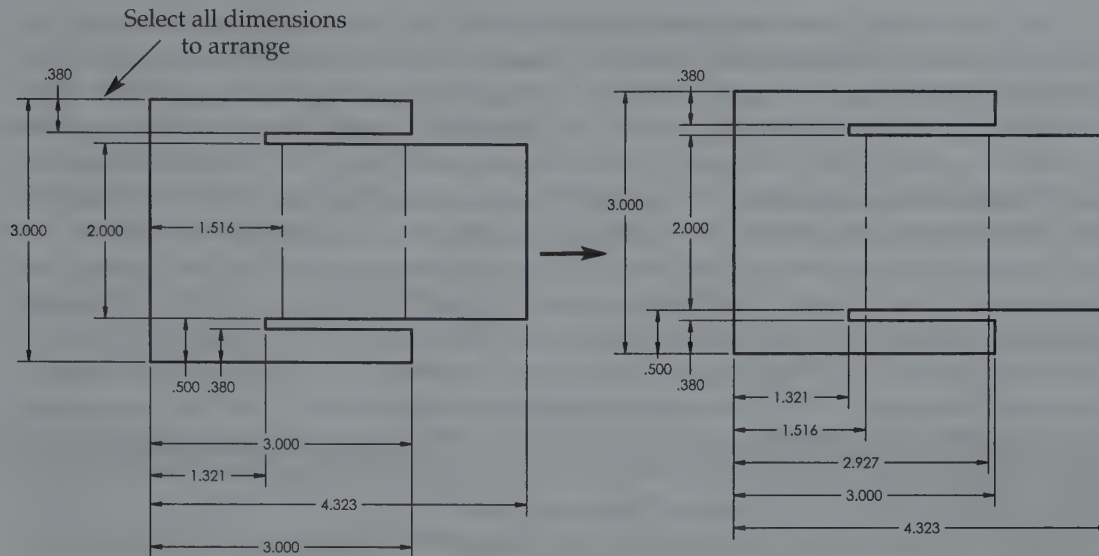
Arrange Dimensions tool functions very differently depending on the type and location of selected dimensions and the way you access the tool.

To arrange the dimensions as shown in **Figure 23-10**, you could access the **Arrange Dimensions** tool, select all the dimensions in the view, and then press [Enter] or right-click and select **Done.** Another option is to select all the dimensions and then access the **Arrange Dimensions** tool to create the arrangement instantly. To arrange the group of horizontal or vertical dimensions only, you could access the **Arrange Dimensions** tool and select the dimensions. Then press [Enter] or right-click and select **Done** to create the arrangement or right-click and pick **Contour Entity** to select a point or object to specify where the first offset originates. If you select the group of horizontal or vertical dimensions before accessing the **Arrange Dimensions** tool, the **Contour Entity** function may be preset, allowing you to pick a point or object.

Exercise 23-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-5.

Editing Model Dimensions

If a design requires a revision, open and edit the model as needed. Dimensions in the drawing update according to the changes. An alternative is to edit model dimensions in a drawing view. Right-click on a model dimension and select **Edit Model Dimension....** The familiar **Edit Dimension** dialog box appears, allowing you to change the value of the parameter. Drawing dimensions associated with the model change as you edit a model, but they do not control model parameters.
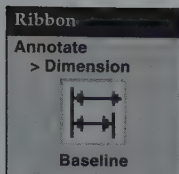
Baseline Dimensioning

You can apply *baseline dimensioning* using the **General Dimension** tool, but Inventor provides **Baseline Dimension** and **Baseline Dimension Set** tools to help apply baseline dimensions. See **Figure 23-11**. The process of creating baseline dimensions is the same regardless of the tool you use. The **Baseline Dimension** tool creates separate dimensions, much like arranging dimensions created using the **General Dimension** tool. The **Baseline Dimension Set** tool creates a group of dimensions that act more like one unit, and includes options for adding and removing dimensions from the set.

Once you access the preferred tool, select the common surface, or *datum*, and then pick each additional object to dimension. Typically, the order in which you select additional features does not matter. However, you must select the common surface first. See **Figure 23-11**. Next, right-click and pick **Continue** and then pick a location to place the dimensions. The first dimension line should snap into position as you near the standard offset. If necessary, you can add dimensions to the group by picking additional features. To redefine the datum, right-click on a different object and select **Make Origin**. Right-click and select **Create** to finish.

Baseline dimensions created using the **Baseline Dimension** tool are independent from other dimensions and function like most dimensions. However, dimensions placed using the **Baseline Dimension Set** tool are grouped and act as a single unit. To add a dimension to the set, right-click on the set, select **Add Member**, and pick the geometry to add. To remove a dimension from the set, right-click on the dimension and select **Delete Member**. To disassociate a dimension from a set without deleting the dimension, right-click on the dimension and select **Detach Member**.

baseline dimensioning: Dimensioning that defines the size and location of each feature in reference to an origin.

datum: A theoretically exact point, axis, or plane from which the location or geometric characteristics of features originate.

Exercise 23-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-6.

Figure 23-11.
Placing baseline dimensions using the **Baseline Dimension** or **Baseline Dimension Set** tool involves the same steps.
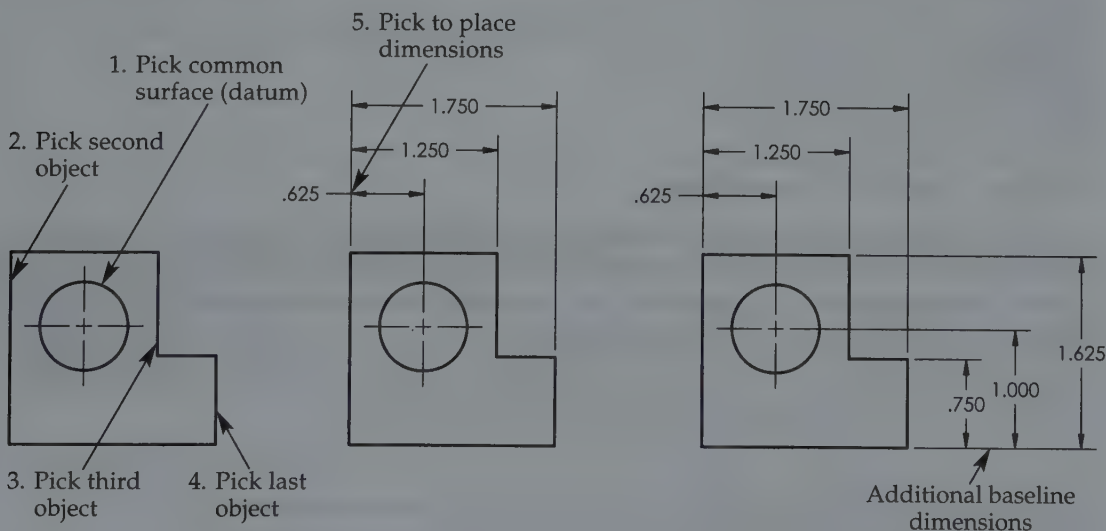
Notes

Drawings usually require *specific notes* and *general notes*. A *leader line (leader)* often connects a specific note to a feature. Inventor provides several tools for adding notes. When possible, use parametric note tools that reference model parameters. Use **Text** and **Leader Text** tools when you cannot extract data from a model, or for general applications.

Access the **Text** tool to place general notes or other text not connected to a leader. The **Text** tool available in the drawing annotation environment functions the same as the **Text** tool available in sketch environments. Pick a point or create a text box, and then use the **Format Text** dialog box to add text. When you return to the graphics window, continue adding text or press [Esc], right-click and select **Done**, or access a different tool to exit.

specific notes: Notes that apply to a specific feature or features on the drawing.

general notes: Notes that apply to the entire drawing and are usually placed together in a standard location such as the lower-left corner, upper-right corner, or near the title block.

> **NOTE**
>
> Right-click on text and select **Edit Text...** to return to the **Format Text** dialog box, or use the **Rotate 90 CW** or **Rotate 90 CCW** options to rotate the text.

leader line (leader): A line drawn at an angle from a specific note to a feature. One end of the leader attaches to a short shoulder that connects it with the note text, and the other end is capped with an arrowhead that touches the feature.

The **General Dimension** tool allows you to dimension circular features with leaders. Additional tools are available for dimensioning hole, chamfer, punch, and bend features. For applications when these tools are not appropriate, access the **Leader Text** tool to create text attached to a leader. See **Figure 23-12**. Select a point or a feature to connect to the leader. Then pick the second point of the leader, which usually defines the start of the shoulder. You can select additional points, but for most applications, the second point is all that is required. Press [Enter] or right-click and select **Continue** to display the **Format Text** dialog box and add text to the leader. When you return to the graphics window, continue adding text or press [Esc], right-click and select **Done**, or access a different tool to exit.

> **NOTE**
>
> The active standard controls default text, leader text, and feature note properties. You have the option of adjusting display characteristics and overriding the default style as needed.

Figure 23-12. Creating a specific note using the **Leader Text** tool.

Pick second and press **[Enter]**

Shoulder forms automatically

32

19

1:1

19

Type the text using the **Format Text** dialog box

Pick first
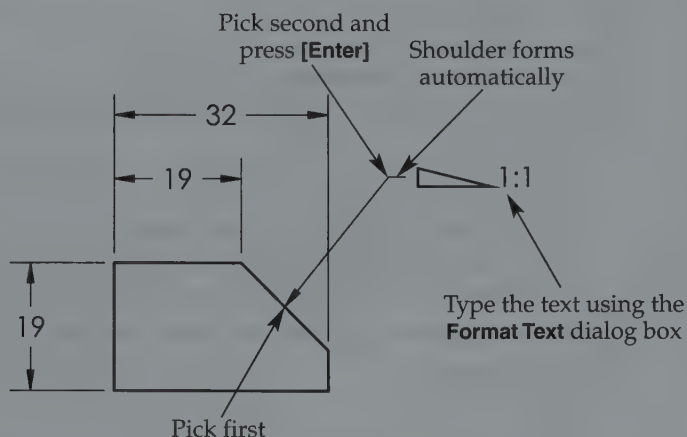
Exercise 23-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-7.

Hole and Thread Notes

Holes are normally dimensioned using diameter dimensions, which include a leader line and dimension value, with the diameter symbol preceding the value, as in Ø1.625. Thread notes commonly connect to the thread representation using a leader line and always follow the same format, as shown in **Figure 23-13**. The **Hole/Thread Notes** tool allows you to extract the specifications of a hole or thread feature.

Select the feature to dimension and pick a location for the note. Press [Esc], right-click and select **Done**, or access a different tool to exit. To edit a hole note using the **Edit Hole Note** dialog box, double-click on the note or right-click on the note and select **Edit Hole Note...**. A common edit is to pick the **Tap Drill** check box to add tap drill data to the note. To apply overrides to the default note, deselect the **Use Default** check box.

Ribbon
Annotate
> Feature Notes

Hole and Thread
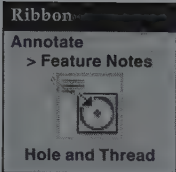
> **NOTE**
>
> To dimension repetitive holes, pick the **Edit Quantity Note** button and specify quantity based on a feature pattern in the model or like holes in the drawing. Then deselect the **Use Default** check box, place the text cursor before the note, and pick the **Quantity Note (#)** button.

Figure 23-13.
Thread note specifications and dimensioning threads using the **Hole/Thread Notes** tool.
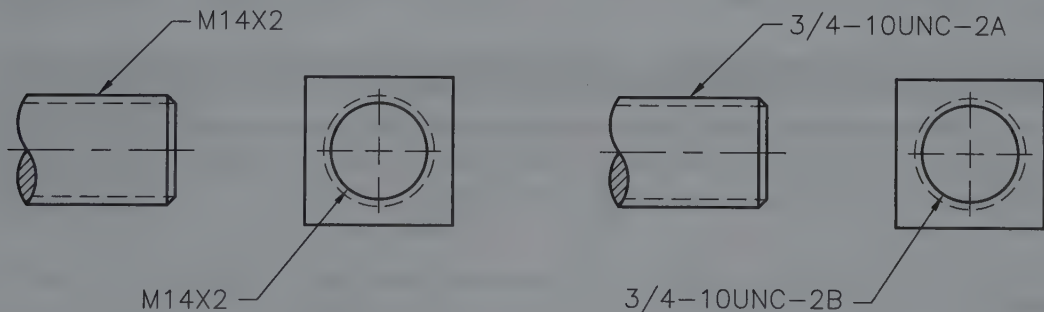
The following format specifies the thread note for Unified screw threads.

3/4-10UNC-2A
 (1)   (2)   (3)   (4)(5)

(1) Major diameter of thread, given as a fraction or number.
(2) Number of threads per inch.
(3) Thread series: UNC = Unified National Coarse, UNF = Unified National Fine.
(4) Class of fit: 1 = large tolerance, 2 = general-purpose tolerance, 3 = tight tolerance.
(5) Thread type: A = external thread. B = internal thread.

The following format specifies the thread note for metric threads.

M14X2
(1) (2)   (3)

(1) M = metric thread.
(2) Major diameter in millimeters.
(3) Pitch in millimeters.

Dimensioning Chamfers

You can use a specific note to dimension a 45° *chamfer* as shown in **Figure 23-14A**. Chamfers at angles other than 45° must display the angle and a linear dimension or two linear dimensions, as shown in **Figure 23-14B**. The **Chamfer Note** tool allows you to extract the specifications of a chamfer feature as a specific note, which is appropriate for 45° chamfers. Select the chamfered surface followed by the adjacent reference edge. Then pick a location for the note. Press [Esc], right-click and select **Done**, or access a different tool to exit. Double-click on the note or right-click on the note and select **Edit Chamfer Note...** to change the note using the **Edit Hole Note** dialog box.

Ribbon

Annotate
> Feature Notes
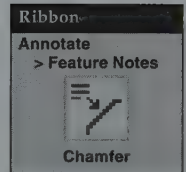
Chamfer

chamfer: A slight surface angle used to relieve a sharp corner.
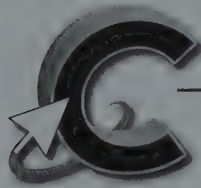
Exercise 23-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-8.

Punch and Bend Notes

The **Punch Notes** tool allows you to extract the specifications of a sheet metal punch feature. Select the punch geometry, representation, or center point to dimension and then pick a location for the note. See **Figure 23-15**. Press [Esc], right-click and select **Done**, or access a different tool to exit. To edit a punch note using the **Edit Punch Note** dialog box, double-click on the note or right-click on the note and select **Edit Punch Note...**.

Ribbon

Annotate
> Feature Notes

Punch
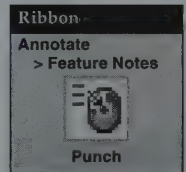
The **Bend Notes** tool allows you to extract the specifications of a sheet metal bend. Select the bend centerline to place a note aligned with the centerline. Continue adding bend notes, or press [Esc], right-click and select **Done**, or access a different tool to exit. You can then drag the note to adjust the position or add a leader if necessary. See **Figure 23-15**. To edit a bend note using the **Edit Bend Note** dialog box, double-click on the note or right-click on the note and select **Edit Bend Note...**.

Ribbon

Annotate
> Feature Notes

Bend
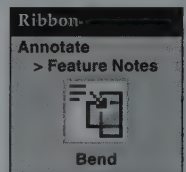
Exercise 23-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-9.

Figure 23-14.
A—Common practices for dimensioning chamfers. You can use the Chamfer Note tool to dimension any chamfer feature, but chamfer notes are only appropriate for 45° chamfers. B—Chamfers other than 45° must display the angle and a linear dimension or two linear dimensions.

A

B
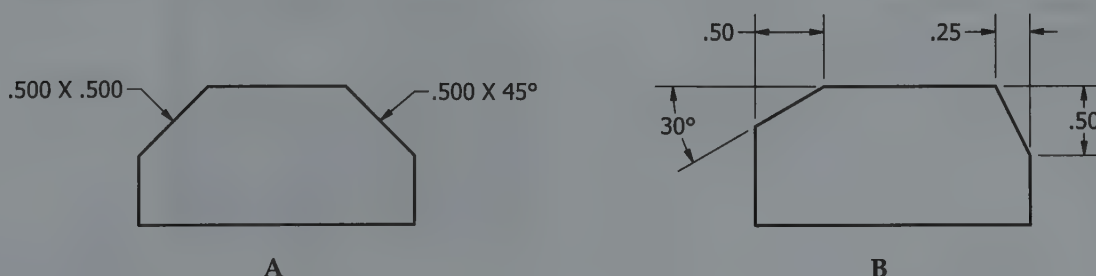
Figure 23-15.
An example of a flat pattern drawing view with punch and bend notes extracted from the model.

Punch note

2X PID67495B
DOWN
90°

Bend note (drag after
placement to form leader)

DOWN 90° R.120

Punch note

PID67654A
DOWN
90°

UP 90° R.120

Bend note
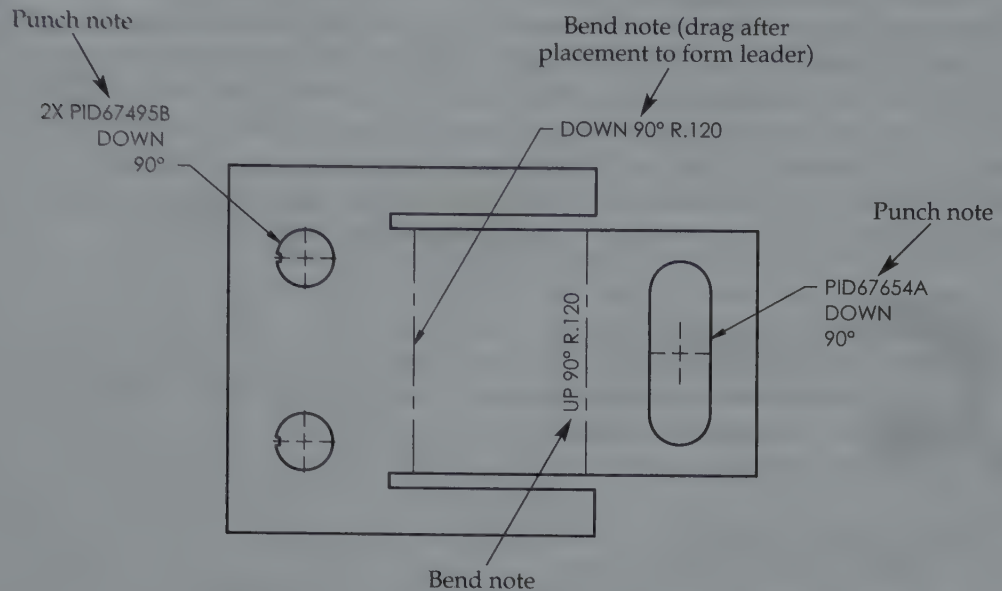
Surface Texture Symbols

Use the **Surface Texture Symbol** tool to specify *surface finish* using a *surface finish symbol*. See **Figure 23-16**. Select a point or feature to assign surface finish information. To locate the symbol at the selection, press [Enter] or right-click and select **Continue**. To connect the symbol to a leader, pick a second point and then press [Enter] or right-click and select **Continue**.

The **Surface Texture** dialog box, shown in **Figure 23-16**, allows you to define the finish characteristics. **Figure 23-17** describes each option. Pick the **OK** button to create the surface finish symbol. Press [Esc], right-click and select **Done**, or access a different tool to exit.
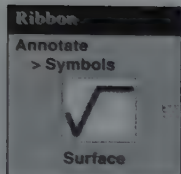
surface finish: The allowable roughness, waviness, lay, and flaws on a surface.

surface finish symbol: A V-shaped symbol that contains values related to surface finish characteristics.

PROFESSIONAL TIP

Drag the grip point that appears when you select an existing surface texture away from the feature to include an extension line or add a leader.

Figure 23-16.
Examples of surface finish specifications creating using the **Surface Texture Symbol** tool. You can connect the surface finish symbol to a leader by selecting a second point.

Removal of material prohibited

Removal of material required

Basic Surface texture symbol

Force tail

Majority

All-around

Surface Texture

Surface Type

Miscellaneous

63 .0005
32 .010
.010 X

B .0005
B'
A' 63   C .010
A 32   C'
E .010   D X   F'

Ok   Cancel

125

125
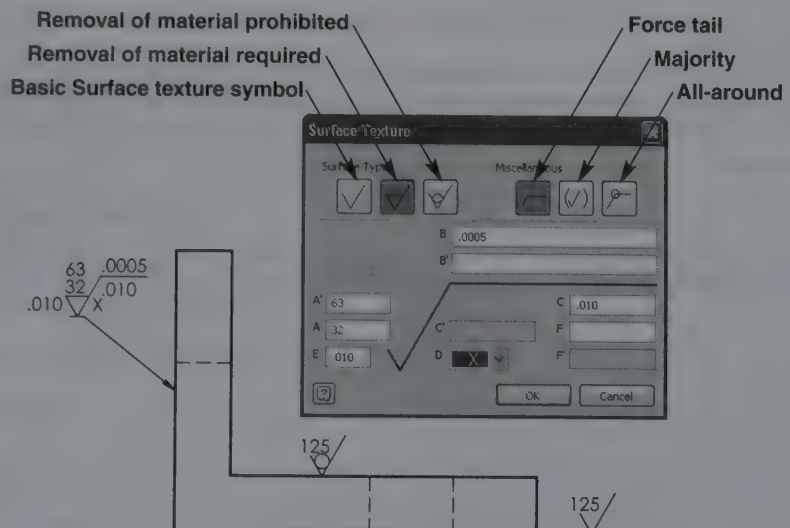
Figure 23-17.
Options available in the **Surface Texture** dialog box.

Item	Description
Surface Type	Select the **Basic Surface texture symbol**, **Removal of material required**, or **Removal of material prohibited** button.
Miscellaneous	Select the **Force tail**, **Majority**, or **All-around** button to add a tail, majority, or all-around symbol.
'	Maximum roughness height, or the roughness average (Ra).
A	Minimum roughness height, when a maximum roughness height is set in the **A'** text box.
B	Waviness of the surface texture.
B'	Waviness width, or pitch, of the surface texture.
C' or C	Roughness width cutoff or sampling length, depending on the application.
D	Machining lay.
F	Roughness other than the Ra specified in the **A'** text box.
F'	Minimum roughness other than the Ra specified in the **F** text box.
E	Machining allowance.

Exercise 23-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-10.

Ordinate Dimensioning

Inventor refers to *rectangular coordinate dimensioning without dimension lines* as *ordinate dimensioning*. Use the **Ordinate Dimension** or **Ordinate Dimension Set** tools to apply ordinate dimensions. See **Figure 23-18**. The **Ordinate Dimension** tool creates separate

Figure 23-18.
Placing ordinate dimensions using the **Ordinate Dimension** or **Ordinate Dimension Set** tool involves similar initial steps. Turn off the visibility of the origin indicator if it is not needed.

Ribbon

Annotate
> Dimension

Ordinate

Ribbon

Annotate
> Dimension

Ordinate Set

Type

[O]
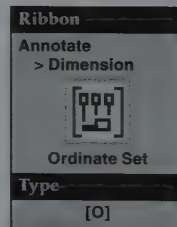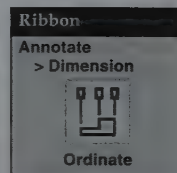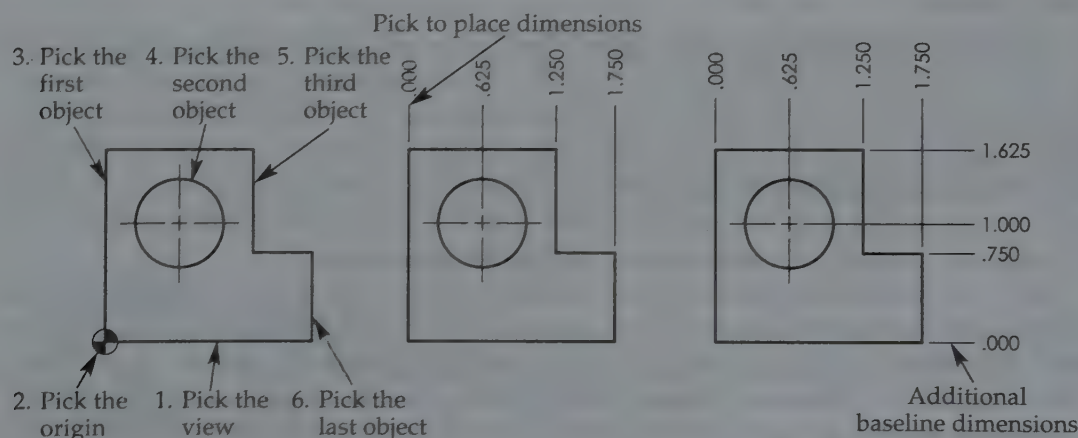
rectangular coordinate dimensioning without dimension lines (ordinate dimensioning): Dimensioning in which dimension values are placed at the end of extension lines, providing coordinates from established planes or the axis of a feature.

dimensions. The **Ordinate Dimension Set** tool creates a group of dimensions that act more like one unit and includes options for adding and removing dimensions from the set.

Once you access the preferred tool, pick a view, and then select the *origin*. You must select a view because each view must have an origin. Then pick each object to dimension. Typically, the order in which you select features does not matter, but remember to select origin geometry to include. See **Figure 23-18**. Next, right-click, pick **Continue**, and pick a location to place the dimensions. Several additional shortcut menu options are available when you are using the **Ordinate Dimension Set** tool. If necessary, you can add dimensions to the group while using either tool by picking additional features. To exit the **Ordinate Dimension** tool, press [Esc], right-click and select **Done**, or access a different tool. To exit the **Ordinate Dimension Set** tool, right-click and select **Create**.

Ordinate dimensions created using the **Ordinate Dimension** tool are independent from other dimensions. However, dimensions placed using the **Ordinate Dimension Set** are grouped and act as a single unit. Right-click on ordinate dimensions created using the **Ordinate Dimension Set** tool to access several options for adding and removing dimensions and for adjusting the order and appearance of the set.

> **NOTE**
>
> Turn off the visibility of the origin indicator by right-clicking on the origin indicator or an ordinate dimension and selecting **Hide Origin Indicator**.

Exercise 23-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-11.

table: An arrangement of rows and columns that organize data to make it easier to read.

tabular dimensioning: A system of ordinate dimensioning in which coordinate dimensions and size dimensions are given in a table and correlated to features on the drawing with a tag.

hole table: A common form of tabular dimensioning that specifies the size and location of holes using a table.

Ribbon
Annotate
> Table
> Hole

Hole Selection

Ribbon
Annotate
> Table
> Hole

Hole View

Ribbon
Annotate
> Table
> Hole

Hole Features

Tables

Tables are commonly used for a variety of drafting applications, including *tabular dimensioning*, a bill of materials or parts list, or a revision history block. Some tools allow you to extract model data to form a table. For example, you can create a hole table from hole features or a bend table from sheet metal bends in a view. You can also create general tables for other applications. The following information focuses on hole, bend, and general tables. You will learn to create parts lists and revision history blocks in Chapter 24.

Hole Tables

The **Hole Table – Selection, Hole Table – View,** and **Hole Table – Selected Feature** tools allow you to extract hole features to a *hole table*. See **Figure 23-19**. The **Hole Table – Selection** tool allows you to select specific hole features in a systematic order. The **Hole Table – View** tool selects all hole features automatically when you pick a view. The **Hole Table – Selected Feature** tool allows you to select a hole, and then adds all holes with the same parameters to the table. Hole table tools do not fully dimension a view. You must still add centerlines, center marks, and other dimensions.

Once you access the preferred tool, pick a view and select the origin (unless you already specified the origin, such as when creating ordinate dimensions). In the **Hole Table – Selection** tool, the next step is to pick each hole feature to include in the table. In the **Hole Table – Selected Feature** tool, the next step is to pick each hole feature to include in the table, but you only need to select one instance of repetitive holes.

Figure 23-19.
A—Using a hole table tool to create a hole table. B—Completing the drawing with center marks and ordinate dimensions. Notice the changes made to the position of the tags. This was done by dragging the tags to a new location.

3 or 4. Place the table, depending on the active tool

1. Pick the view

3. Pick features or place the table, depending on the active tool

2. Pick the origin

HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	0.16	0.16	Ø3 THRU
A2	3.62	0.16	Ø3 THRU
A3	3.62	0.83	Ø3 THRU
A4	0.16	1.81	Ø3 THRU
A5	2.60	1.81	Ø3 THRU
B1	2.28	0.39	Ø6 THRU ⌴Ø7 ▼ 5
C1	0.98	0.59	Ø6 THRU ⌴Ø10 ▼ 5
D1	1.97	1.18	Ø20 ▼ 15
E1	0.71	1.38	M5x0.8 - 6H ▼ 6

A

HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	0.16	0.16	Ø3 THRU
A2	3.62	0.16	Ø3 THRU
A3	3.62	0.83	Ø3 THRU
A4	0.16	1.81	Ø3 THRU
A5	2.60	1.81	Ø3 THRU
B1	2.28	0.39	Ø6 THRU ⌴Ø7 ▼ 5
C1	0.98	0.59	Ø6 THRU ⌴Ø10 ▼ 5
D1	1.97	1.18	Ø20 ▼ 15
E1	0.71	1.38	M5x0.8 - 6H ▼ 6

B

Inventor selects all additional holes of the same type. Both tools require that you select holes in a systematic order beginning with the first hole to list. Once you select all of the holes to include, right-click and pick **Create** to attach the table to the cursor. In the **Hole Table – View** tool, Inventor selects all hole features in the model and attaches the table to the cursor. Pick a location on the sheet to create the table and tags.

The appearance of the table, tags, and feature note properties are initially set according the selected origin and features and the active standard. You can adjust most display characteristics and override the default style. Select, double-click, or use the extensive list of shortcut menu options to make changes. Common examples of adjusting a tag include dragging the tag to a new position or away from the hole to connect the tag to a leader. You can also hide tags and edit tags using the **Format Text** dialog box. Common examples of adjusting the table include dragging cell edges to change row and column spacing and re-sorting table contents when the model and drawing change. You can also split a long table, edit the contents of a cell, add or remove rows and columns, and export the table as a .txt or .csv file. Explore each option as needed to control the appearance of the table and data provided by the drawing.

Exercise 23-12

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-12.
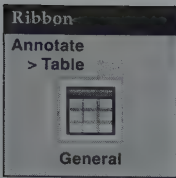
bend table: A table that indicates the location and parameters of sheet metal bends.

Bend Tables

Access the **Table** tool to extract sheet metal bend features to a *bend table*. See **Figure 23-20**. Select a sheet metal view containing bends to acquire model data. You have the option of selecting a different source file and, in some cases, a different set of data to create a different table. The default bend table includes the bend ID, based on the sequence specified in the model, and the direction, angle, and radius. Pick the **Columns Chooser** button to make changes to the available columns and column order. The **Bend ID** area allows you to use numbers or letters to indicate each bend and add a prefix if necessary. Pick the **OK** button and select a location for the table. You can make changes to bend tags and the bend table using techniques similar to those for modifying a hole table.

General Tables

The **Table** tool also allows you to create blank tables without picking a view or referencing drawing or model data. Specify the number of columns in the **Columns** text box and the number of data rows in the **Data Rows** text box. Then pick the **OK** button and select a location for the table. See **Figure 23-21A**. You are responsible for adding content to the table. You can also adjust most display characteristics and override the default style.

Most table editing options are controlled in the **Edit Table** dialog box, shown in **Figure 23-21B**. Access this dialog box by double-clicking on the table or right-clicking on the table and selecting **Edit**. Type values in the cells to add content to the table. The **Edit Table** dialog box includes several buttons and settings for controlling table display and adding or removing data. Right-click on a column, row, or cell to access context-sensitive options. Explore each option as needed to control the appearance of the data.
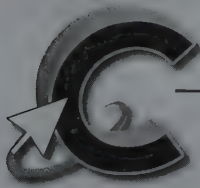
Exercise 23-13

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-13.

Figure 23-20.
An example of a drawing with bend parameters extracted from the model to form a bend table.

Pick to add, remove, or create columns

Selected view

TABLE			
BEND ID	BEND DIRECTION	BEND ANGLE	BEND RADIUS
1	DOWN	90	.25
2	DOWN	90	.25
5	DOWN	90	.25
3	UP	90	.25
4	UP	90	.25
6	DOWN	90	.25

Bend order tags
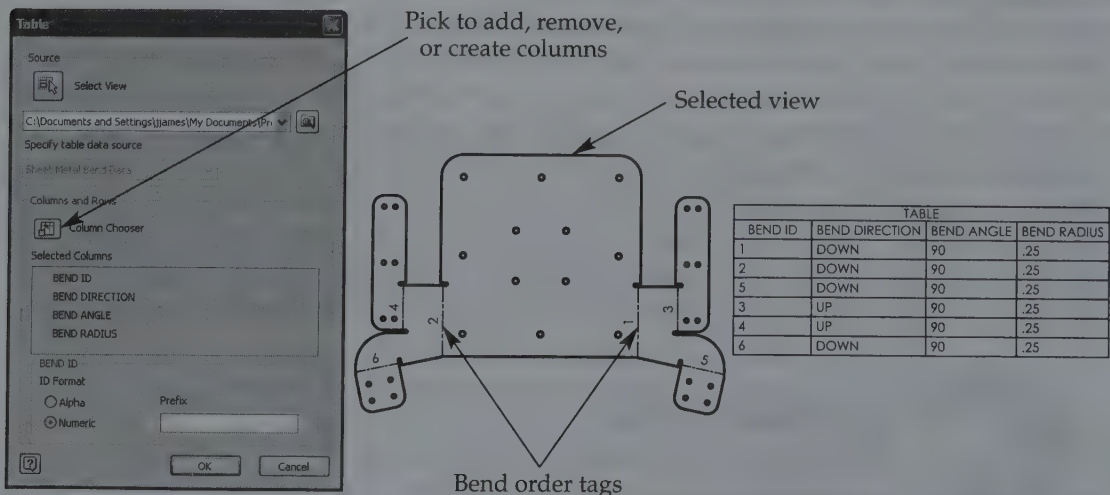
Figure 23-21.
A—Creating a general table using the **Table** tool. B—Using the **Edit Table** dialog box and the many adjustment and control options to finalize a general table.

A

Pick to access the **Table Columns Chooser** dialog box

Select to insert a row

Select to remove a row

Pick to access the **Sort Table** dialog box

Pick to export as a .txt or .csv file

Pick to override the table layout using the **Table Layout** dialog box

Type values in cells, or stretch the size of columns or rows

B
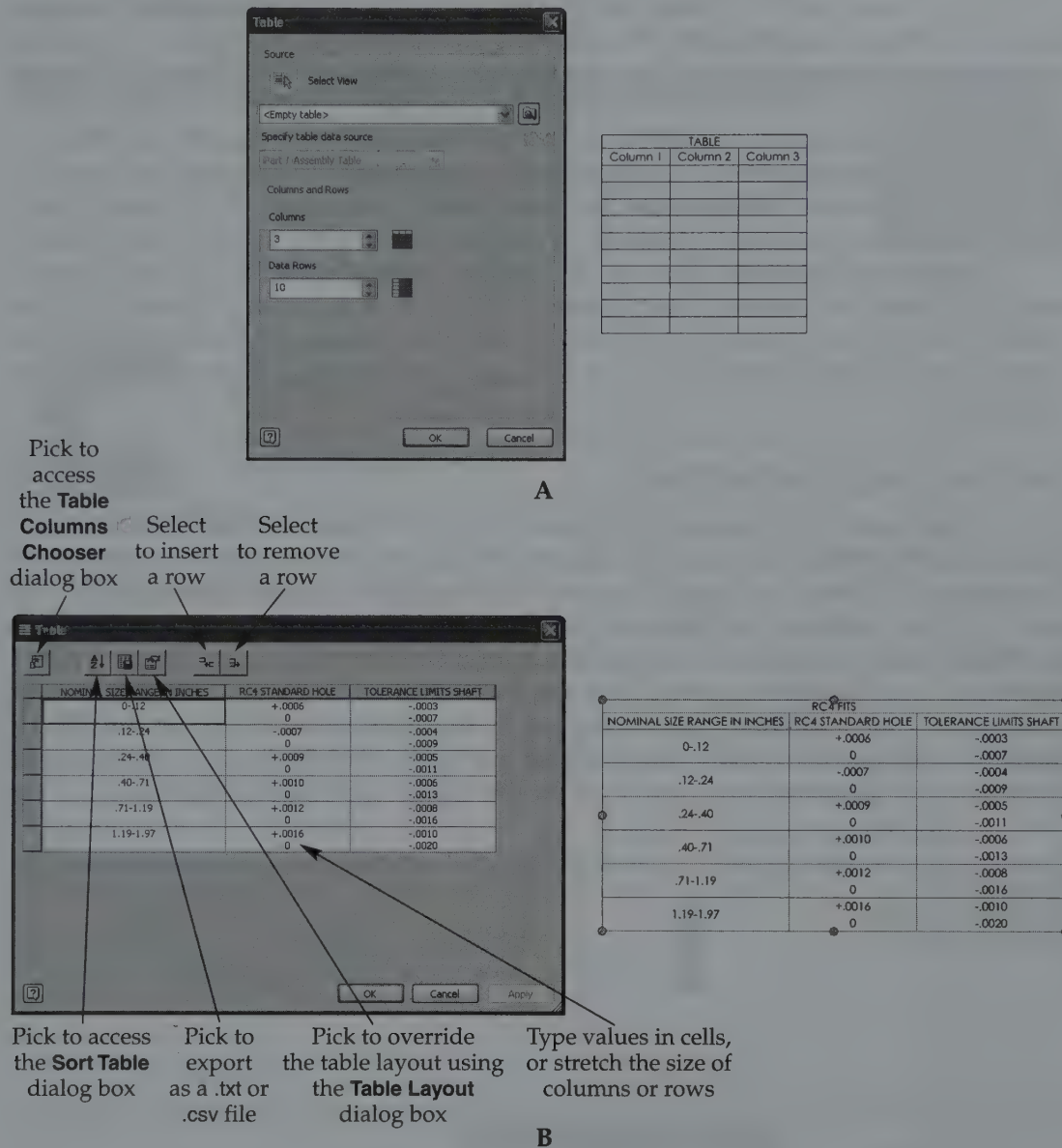
Supplemental Material

GD&T

For information about using *geometric dimensioning and tolerancing (GD&T)* tools, go to the Student Web site (www. g-wlearning.com/CAD), select this chapter, and select **GD&T**. For complete coverage of GD&T, refer to *Geometric Dimensioning and Tolerancing*, by David A. Madsen, published by Goodheart-Willcox Company, Inc.

geometric dimensioning and tolerancing (GD&T): The dimensioning and tolerancing of individual features of a part in which the permissible variations relate to characteristics of form, profile, orientation, runout, or the location of features.

Ribbon
Manage
> Define
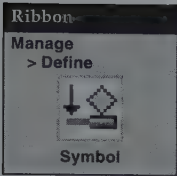
Symbol

You can create your own drawing symbols for applications when default symbols are unavailable or inappropriate. For example, your drawings may require company- or industry-specific symbols, such as those shown in **Figure 23-22**. Access the **Create New Symbol** tool to develop a custom symbol. Use sketch tools as you do in other sketch environments to create a symbol. Finish the sketch to create the symbol and display the **Sketched Symbol** dialog box. See **Figure 23-23A**. Type the symbol name in the **Name** text box and pick the **Save** button to create the symbol. The **Discard** button deletes the sketch and cancels the tool.

The **Sketched Symbols** folder in the **Drawing Resources** folder in the browser contains all sketched symbols. See **Figure 23-23B**. To insert a symbol, double-click the symbol name in the browser or right-click on the symbol name and select **Insert**. Then pick to locate symbols. Press [Esc], right-click and select **Done**, or access a different tool to exit.

User Defined Symbol Tool

Ribbon
Annotate
> Symbols

User

Another method for placing symbols is to access the **User Defined Symbol** tool to display the **Symbols** dialog box. See **Figure 23-23C**. Select the symbol to insert from the **Symbols** list box. To change the size of the symbol, pick the **Scale** button and specify a scale factor in the **Scale** text box. To change the angle of the symbol, pick the **Rotate** button and specify a rotation angle in the **Rotate** text box. Check **Symbol Clipping** to trim dimensions away from the symbol when you insert the symbol over dimensions. Check **Static** to remove the ability to rotate or resize the symbol. Check **Leader** to add a leader to the symbol. Toggle the visibility of the leader by selecting or deselecting the **Visibility** check box. Pick the **OK** button to place the symbol.

Figure 23-22.
Examples of custom drawing symbols.
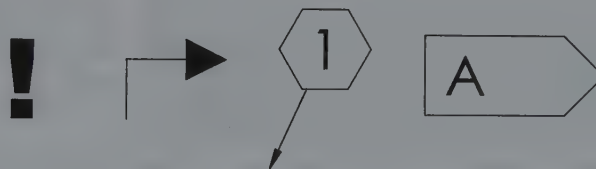
Figure 23-23.
A—The **Sketched Symbol** dialog box. B—Symbols contained in the **Sketched Symbols** folder. C—The **Symbols** dialog box.
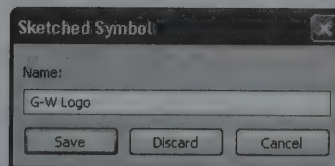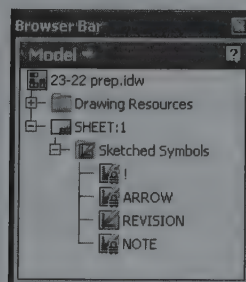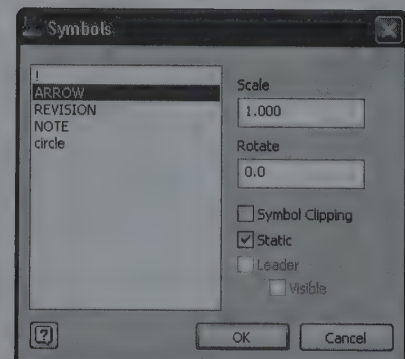
To locate the symbol at the selection without a leader, press [Enter] or right-click and select **Continue**. To connect the symbol to a leader, pick points as needed, and then press [Enter] or right-click and select **Continue**. Press [Esc], right-click and select **Done**, or access a different tool to exit.

NOTE

You can rotate a non-static symbol inserted using the **Symbols** dialog box by selecting the symbol and dragging the blue grip to the desired angle. Scale a non-static symbol by dragging one of the yellow dots. Use the green grip to move a symbol.


Exercise 23-14

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 23-14.

Chapter Test

1. What is a dimension?
2. Briefly describe model dimensions.
3. Describe drawing dimensions.
4. Describe the function of the **Retrieve all model dimensions on view placement** check box on the **Drawing** tab of the **Options** dialog box and the **All Model Dimensions** check box on the **Display Options** tab of the **Drawing View** dialog box.
5. Explain what happens when you right-click on a view and select **Automated Centerlines...**.
6. Define *baseline dimensioning.*
7. Explain the use of specific notes on a drawing.
8. Define *general notes* and identify where they appear on a drawing.
9. Describe the use of leader lines.
10. Explain how holes are normally dimensioned, and include an example of hole dimension text.
11. Define *chamfer.*
12. How are chamfers at angles other than 45° dimensioned?
13. Define *surface finish.*
14. Describe a surface finish symbol.
15. Briefly describe rectangular coordinate dimensioning without dimension lines and give the name used by Inventor for this type of dimensioning.
16. What is the origin?
17. Describe tabular dimensioning.
18. Briefly describe a hole table.
19. What is the basic function of a bend table?
20. Explain how to connect a symbol to a leader and how to place a symbol without a leader.

Problems

Instructions:

- *Open the specified drawing file and save the file using the given name.*
- *Create or adjust drawing views as needed.*
- *Dimension the drawings exactly as shown. Remove or modify existing dimensions as shown.*
- *Update the title block and general note content as needed.*

1. **Title:** DIMENSIONS
   File: EX23-1.idw
   Save as: P23-1.idw

2. **Title:** SWIVEL BOLT
   File: P22-1.idw
   Save as: P23-2.idw

.783

.500

.025 X 45°

60°

Ø.500

Ø.250

1/4-20 UNC - 2A

NOTES:

1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES (IN.) TOLERANCES:	APPROVALS	DATE	MADSEN DESIGNS INC.				
.X ±.1	DRAWN DPM						
.XX ±.01	CHECKED DAM						
.XXX ±.005 ANGULAR: ±10' FINISH: 62µIN.	APPROVED DAM		TITLE SWIVEL BOLT				
THIRD ANGLE PROJECTION	MATERIAL SAE 4140						
	FINISH ALL OVER		SIZE A	CAGE CODE	DWG NO. IAA-009-02		REV 0
	DO NOT SCALE DRAWING		SCALE 2:1			SHEET 1 OF 1	

3. **Title:** C-CLAMP SWIVEL
   File: P22-4.idw
   Save as: P23-3.idw

Ø9.5
Ø6.35
10
3.2
Ø9.5
3.7
Ø19
SECTION A-A

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN MILLIMETERS TOLERANCES:	APPROVALS	DATE	**G-W** Goodheart-Willcox			
X ±0.5 .XX ±0.05 .XXX ±0.005 ANGULAR: ±1° FINISH: 3.2μm	DRAWN DPM					
	CHECKED DPM		TITLE			
	APPROVED DPM		C-CLAMP SWIVEL			
THIRD ANGLE PROJECTION	MATERIAL SAE 1035					
	FINISH ALL OVER		SIZE A3	CAGE CODE	DWG NO IAA-001-02	REV 0
	DO NOT SCALE DRAWING		SCALE 2:1		SHEET 1 OF 1	

4. **Title:** C-CLAMP SWIVEL
   File: P22-5.idw
   Save as: P23-4.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
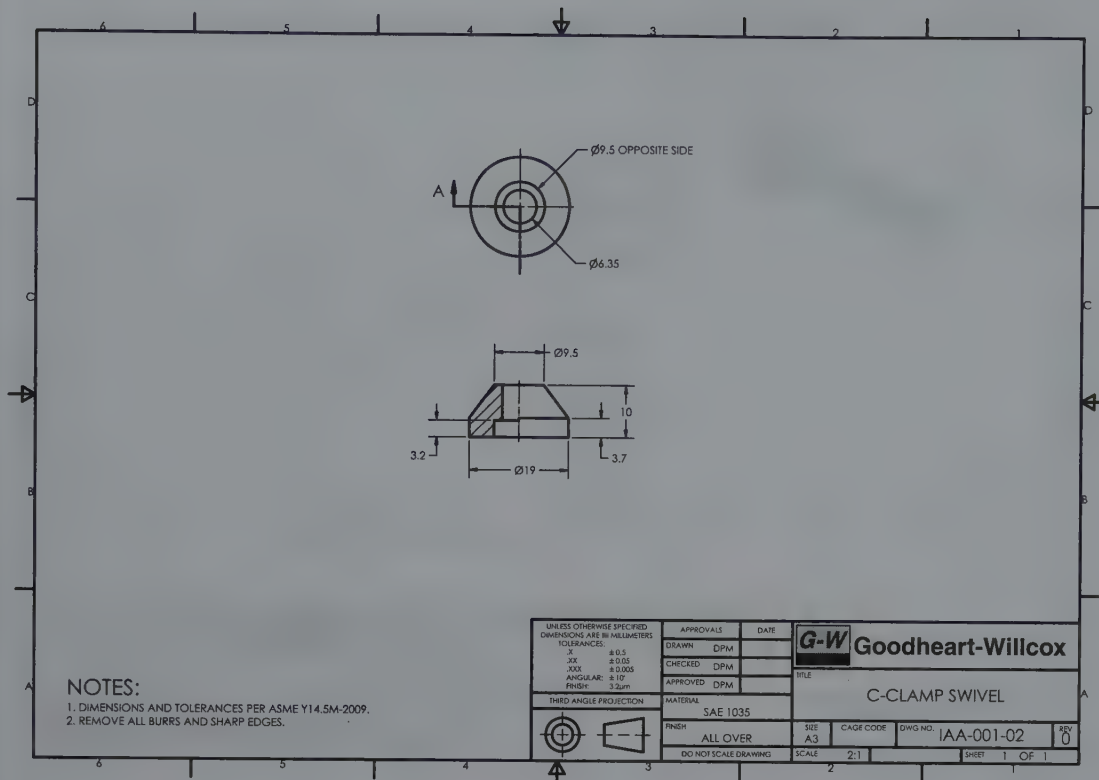
5. **Title:** HANDLE
   File: P22-11.idw
   Save as: P23-5.idw

SECTION A-A

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
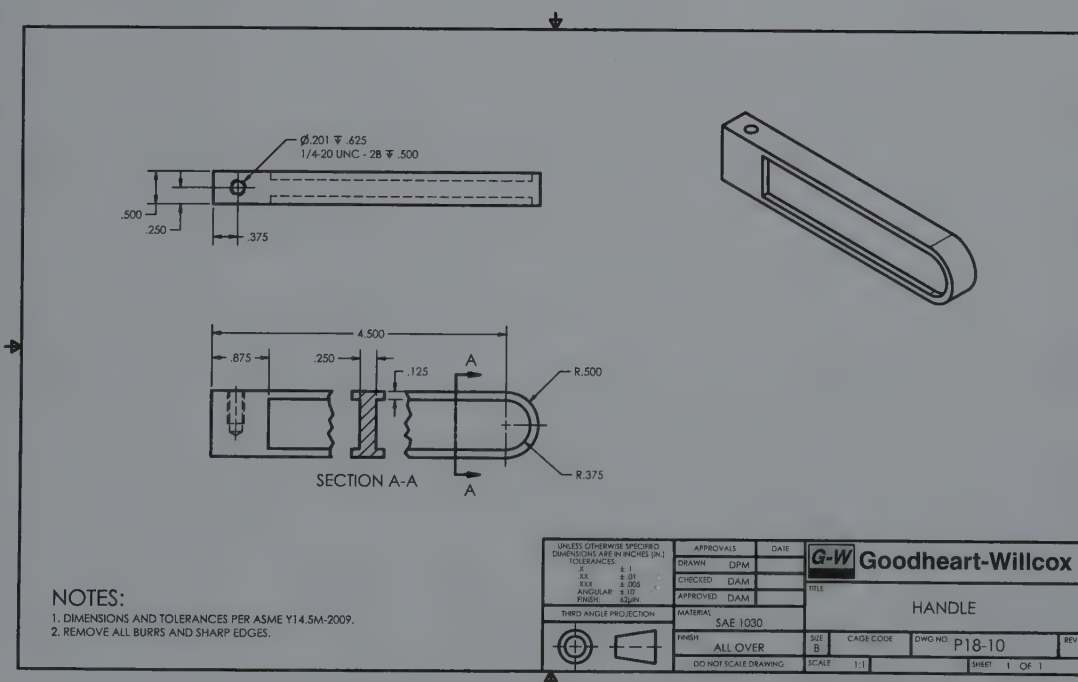
6. **Title:** BRACKET
   File: P22-3.idw
   Save as: P23-6.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN MILLIMETERS TOLERANCES: .X ±0.5 .XX ±0.05 .XXX ±0.005 ANGULAR: ±10' FINISH: 3.2μm	APPROVALS	DATE	*G-W* Goodheart-Willcox
	DRAWN DPM		
	CHECKED DPM		TITLE
	APPROVED DAM		BRACKET
THIRD ANGLE PROJECTION	MATERIAL C6061		
⊕ ⊏	FINISH ALL OVER	SIZE A3	CAGE CODE	DWG NO. IAA-012-01	REV 0
	DO NOT SCALE DRAWING	SCALE 1:1		SHEET 1 OF 1

7. **Title:** SCREW
   File: P22-7.idw
   Save as: P23-7.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN MILLIMETERS TOLERANCES: .X ±0.5 .XX ±0.05 .XXX ±0.005 ANGULAR: ±10' FINISH: 3.2μm	APPROVALS	DATE	*G-W* Goodheart-Willcox
	DRAWN DPM		
	CHECKED DPM		TITLE
	APPROVED DPM		SCREW
THIRD ANGLE PROJECTION	MATERIAL SAE 1035		
⊕ ⊏	FINISH ALL OVER	SIZE A	CAGE CODE	DWG NO. IAA-001-02	REV 0
	DO NOT SCALE DRAWING	SCALE 2:1		SHEET 1 OF 1

Problems – Chapter 23

8. **Title:** CLIP
   File: P22-10.idw
   Save as: P23-8.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

9. **Title:** CENTERS
   File: EX23-3.idw
   Save as: P23-8.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

10. **Title:** BENT PULLER
    File: P22-6.idw
    Save as: P23-10.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
3. R7.500 BEFORE BEND.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES (IN.) TOLERANCES.				
APPROVALS	DATE	MADSEN DESIGNS INC.		
DRAWN DPM				
CHECKED DAM		TITLE		
APPROVED DAM		BENT PULLER		
MATERIAL SAE 51410				
FINISH ALL OVER		SIZE B	CAGE CODE	DWG NO. IAA-010-01	REV 0
DO NOT SCALE DRAWING	SCALE 1:1		SHEET 1 OF 1	

11. **Title:** SUPPORT BLOCK
    File: EX23-5.idw
    Save as: P23-11.idw

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

GOODHEART-WILLCOX

SUPPORT BLOCK

TITANIUM

ALL OVER

IAA-003-01
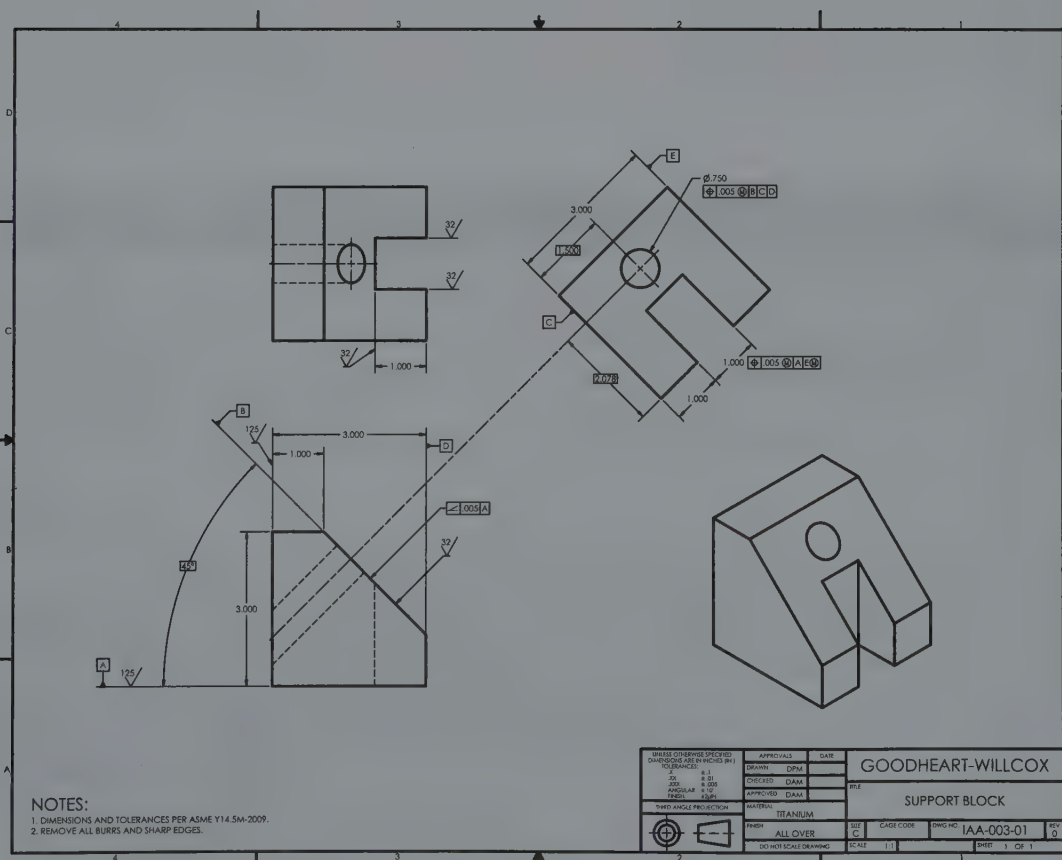
12. **Title:** SLIDER HINGE CONNECTOR
    File: P22-2.idw
    Save as: P23-12.idw

5X Ø.197 THRU
⌴ Ø.394 ▼ .197
⊕ Ø.005 Ⓜ A B C Ⓜ

4X .541 (=2.165)
[.541]
.886

.984

.492

B

.394   .394   .394   .394
.886
1.772
2.657
3.543
3.937

2X R.394

.984

8X R.197
8X R.394   1.969   8X Ø.394
                              ⊕ Ø.005 Ⓜ A B
                              ⊕ Ø.005 Ⓜ

.394

A

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES (IN.) TOLERANCES:		APPROVALS	DATE	MADSEN DESIGNS INC.			
.X ± .1 .XX ± .01 .XXX ± .005 ANGULAR: ± 10' FINISH: 62μIN.		DRAWN	DPM				
		CHECKED	DAM	TITLE			
		APPROVED	DPM	SLIDER HINGE CONNECTOR			
THIRD ANGLE PROJECTION		MATERIAL TITANIUM					
		FINISH ALL OVER		SIZE B	CAGE CODE	DWG NO. IAA-018-01	REV 0
		DO NOT SCALE DRAWING		SCALE 1:1		SHEET 1 OF 1	

Assembly, Weldment, and Multiple-Sheet Drawings

Learning Objectives

After completing this chapter, you will be able to do the following:

✓ Create assembly and weldment drawing views.
✓ Add a parts list and balloons.
✓ Place welding symbols and weld annotations.
✓ Use multiple sheets in a drawing file.
✓ Document engineering changes.
✓ Print and plot drawings.

This chapter explores the process of creating assembly and weldment drawings, which are usually included in a set of *working drawings*. Multiple sheets allow you to prepare a set of drawings in a single drawing file. You will also print drawings and use revision history tools to document engineering changes.

> **working drawings:**
> A set of drawings that includes drawings for each component and an assembly or weldment drawing with a parts list.

Assembly Views

The tools and options for creating an *assembly drawing* are the same as those for a part drawing, except that you reference an assembly or presentation model. Prepare a sheet by adjusting the sheet size and other parameters and inserting a border and title block. Use the **Base View** tool to add the primary assembly view. Then add additional views, such as sections, as needed to document the product.

> **assembly drawing:**
> A two-dimensional representation of an assembly.

Base Views

Access the **Base View** tool to create a base assembly view using the **Drawing View** dialog box. See **Figure 24-1**. Use the options in the **Component** tab to select an assembly file (*.iam), shown in **Figure 24-1A**, to display an assembly view. Select a presentation file (*.ipn), shown in **Figure 24-1B**, to create an *exploded assembly drawing*. You can select a currently open file from the **File** drop-down list. To reference a model that is not open, pick the **Browse** button to locate and select the file. The file then appears in the **File** drop-down list.

> **Ribbon**
> **Place Views**
> **> Create**
>
> **Base**
>
> **Shortcut**
> Base View...
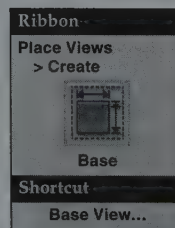
> **exploded assembly drawing:**
> A representation of an assembly in an unassembled configuration.

Figure 24-1.
A—Using the **Base View** tool to create a general assembly view by referencing an assembly model. B—Using the **Base View** tool to create an exploded pictorial assembly view by referencing a presentation model.

A

B
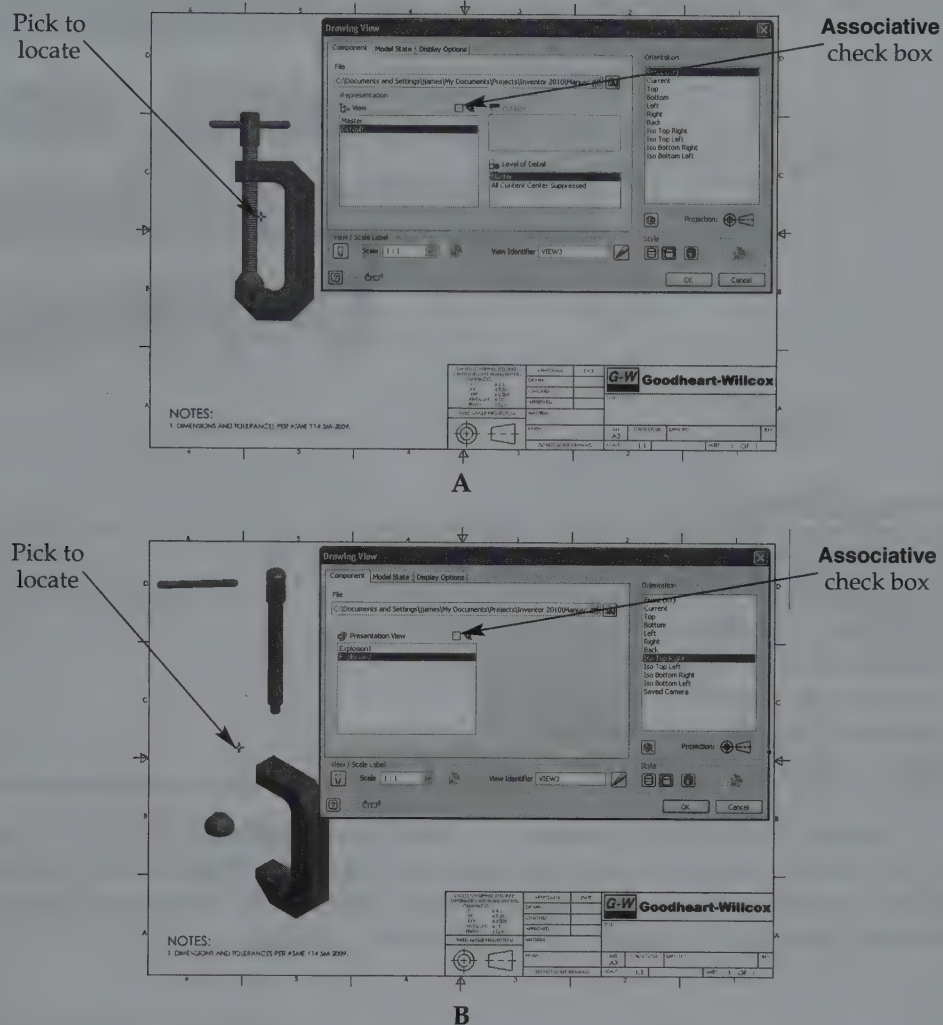
The content of the **Component** tab varies depending on the model you select. When you select an assembly, the **Representation** area appears, allowing you to pick the appropriate design view, position, and level-of-detail representation. Pick the **Associative** check box to update the drawing view when you make changes to the design view representation in the model. When you select a presentation file, the **Presentation View** list box appears, allowing you to select the appropriate presentation view. Pick the **Associative** check box to update the drawing view when you make changes to the presentation view.

When you select an assembly or presentation file, the **Model State** tab provides a **Reference Data** area that allows you to select how to treat *reference data* in the base view. The **Display Options** tab contains multiple check boxes corresponding to items in the selected model that you can display in the base view. The **View Justification** drop-down list allows you to override the default position of the base view, as specified in the **Drawing** tab of the **Options** dialog box. The **Section Standard Parts** drop-down lists allow you to override the standard part sectioning in the base view, as specified in the **Drawing** tab of the **Options** dialog box.

reference data:
An assembly component included for reference purposes only. Reference data is given a reference status in the bill of materials.

The **Orientation**, **Scale**, **Label**, and **Style** areas function the same as when you create a base part view. For a typical multiview drawing, such as a *general assembly drawing*, *working drawing*, *erection assembly drawing*, or exploded multiview drawing, display a standard view as the base view. You can then project additional views as needed. For a *pictorial assembly drawing*, select an isometric view, or create a custom view if necessary. To place the base view, pick a location in the graphics window or pick the **OK** button.

general assembly drawing: An assembly in multiview format that shows the fully assembled product.

working drawing (detail assembly drawing): An assembly drawing that includes details of each assembly component on the same page as the assembly.

erection assembly drawing: An assembly drawing that includes dimensions and fabrication specifications.

pictorial assembly drawing: An assembled or exploded assembly drawing that shows a pictorial view of an assembly, such as a single isometric view, instead of multiple views.

PROFESSIONAL TIP

The **Hidden Line Removed** view style option is commonly used for assembly drawings, but you must display items using options in the **Display Options** tab to represent features, such as threads.
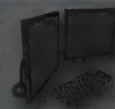
NOTE

Right-click on an assembly view in the browser or graphics window and select **Apply Design View...** to open the **Apply Design View Representation** dialog box, where you can change the design view representation. Expand the assembly drawing view node in the browser to access assembly components and features in the drawing environment.

Exercise 24-1

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-1.

Exercise 24-2

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-2.

Adding Views

Once you place the base assembly view, add views using the same tools and general processes used to add views to a part drawing. Include additional base views, project views, and create auxiliary, section, detail, and break views as needed. To create a single sectioned assembly view, place a base view where the cutting plane will be placed, outside of the sheet. Then cut the base view and position the sectional view on the sheet. See **Figure 24-2**. To display certain components without sectioning, such as the pin shown in **Figure 24-2**, right-click on the component in the browser and select **None** from the **Section Participation** cascading submenu. Adjust hatch patterns as needed to differentiate components and material.

NOTE

You can control the sectioning characteristics of standard parts in an assembly section view by modifying the **Section Standard Parts** variable on the **Display Options** tab of **Drawing View** dialog box.

Figure 24-2.

Creating a single sectional assembly. Notice that section view participation has been removed from the pin, showing the pin in the appropriate unsectioned format.

Top view used to create the section view is placed off the sheet

Turn off label visibility

Sectioning disabled Adjust hatch pattern
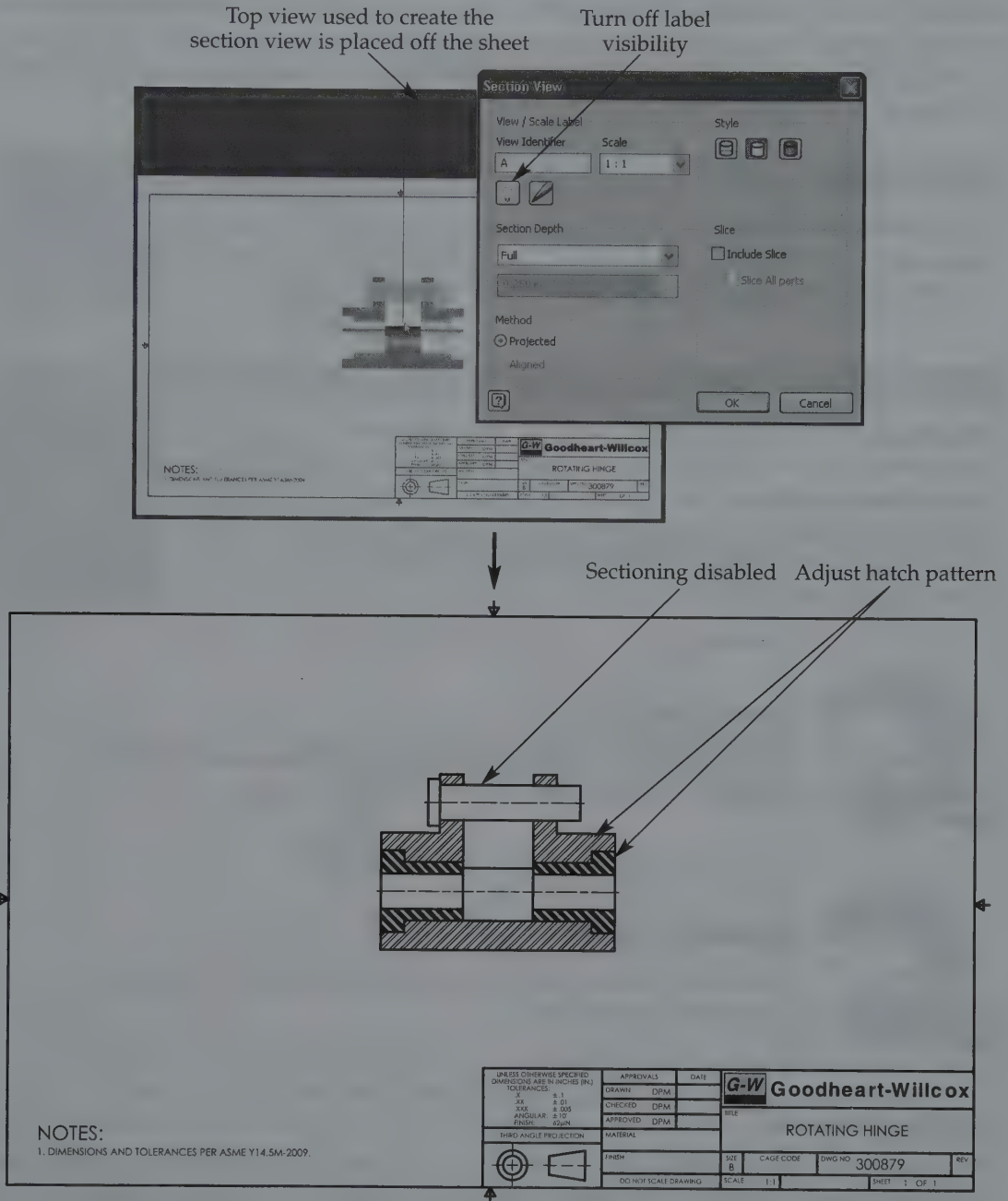
alternate position drawing views (multiple position drawing views): Drawing views that document product design by showing an assembly in more than one position on a single sheet, typically using phantom lines to distinguish alternate positions.

Ribbon

Place Views > Create

Overlay

Shortcut

Create View > Overlay View...

Overlay Views

The **Overlay View** tool allows you to create *alternate position drawing views,* also known as *multiple position drawing views,* using the **Overlay View** dialog box. See **Figure 24-3**. An overlay drawing view references a positional representation in the assembly model. Select a view if you have not already, and then pick a positional representation from the **Positional Representation** drop-down list.

Use the **View Representation** drop-down list to select a design view representation to use for the overlay view. Pick the **Associative** check box to update the drawing view when you make changes to the design view representation in the model. Apply a name different from the positional representation name by deselecting the **Use Positional Rep Name** check box and entering a new name in the **Label** text box. Use the **Toggle**

Figure 24-3.
An example of creating an alternate position drawing by adding overlay views to a base view. Use the **Overlay View** dialog box to reference positional representations in the assembly model.
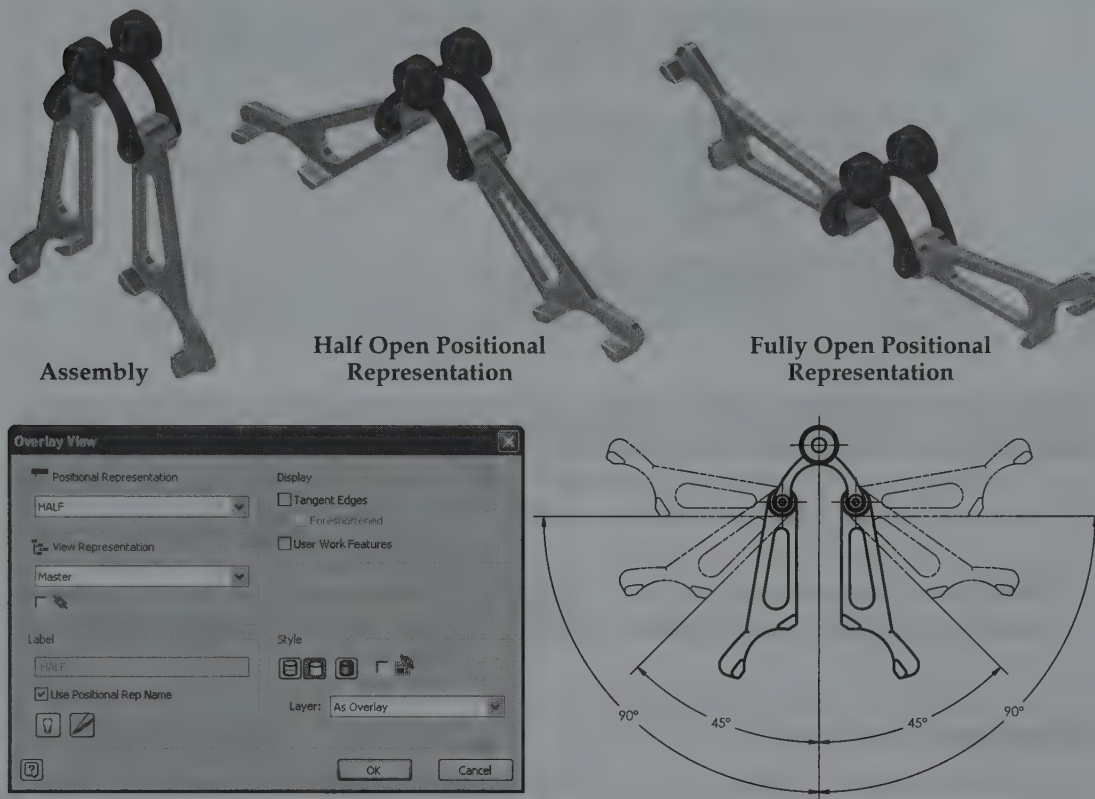
Assembly   Half Open Positional   Fully Open Positional
                     Representation   Representation

Label Visibility button to display the view label, and use the **Edit View Label** button to make changes to the default label.

Specify the style of view geometry in the **Style** area. The **Style from Base** check box sets the overlay view style according to the selected assembly view style. Use the **Layer** drop-down list to select the layer assigned to overlay view geometry. The **As Overlay** layer option is appropriate for most applications, but you can pick the **As Parts** option to use the same layers that are applied to drawing views. Pick the **OK** button to create the overlay view.

Exercise 24-3

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-3.

Parts List

An assembly drawing typically includes a *parts list*. See **Figure 24-4**. A parts list provides different information depending on the application and company standards. It may include columns that specify item number, quantity, part number, drawing number, description, and material. Additional product- or company-specific items may also appear, such as vendor information and other purchase part content. Each row identifies a different component.

parts list: A record of assembly components with item numbers keyed to components in the drawing views.

Figure 24-4.
An assembly drawing typically includes a parts list that is keyed to components to indicate assembly and component specifications.

ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
3	1	300879-03	ROTATING HINGE PIN	SAE 51409
2	2	300879-02	ROTATING HINGE BUSHING	NYLON
1	1	300879-01	ROTATING HINGE HOUSING	SAE 51410

PARTS LIST

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
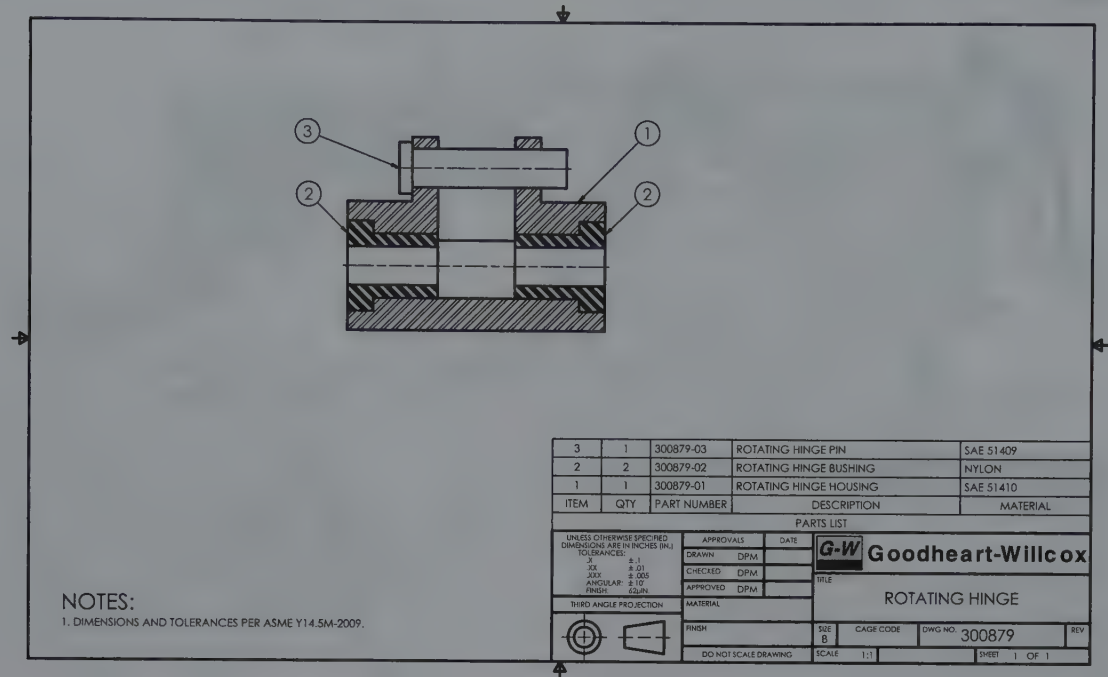
Bill of Materials

bill of materials (BOM): The model version of a parts list; allows you to control assembly and component specifications required to manufacture the assembly.
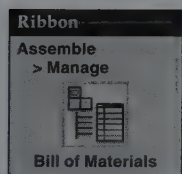
The **Parts List** tool allows you to extract information from an assembly's *bill of materials (BOM)* to create a parts list. The BOM references data from the assembly and each component. For example, the BOM calculates a quantity of 3 if an assembly includes three of the same components, and displays the part number assigned in the component properties. Adjust the BOM and confirm assembly properties and parameters before adding a parts list. You can access the BOM from within an assembly file using the **Bill of Materials** tool. To access the BOM from within the drawing, right-click on an assembly view or a parts list and select **Bill of Materials....** **Figure 24-5A** shows the BOM associated with the drawing and corresponding assembly model shown in **Figure 24-4**.

You can adjust the contents and format of the parts list independently of the **BOM**, but for most applications, the information that appears in the **BOM** should be the same as that in the parts list. The **BOM** is a convenient location to edit parts list data parametrically. For example, to change a part number, enter a new value in the appropriate text box in the **Part Number** column of the **BOM**. The part number updates in the model file, which you can observe in the **iProperties** dialog box. An alternative is to make changes directly in the component files. **Figure 24-5B** describes basic options available in the **BOM**.

Ribbon
Annotate
> Tables

Parts List

Ribbon
Assemble
> Manage

Bill of Materials
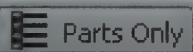
> **NOTE**
>
> The **BOM** includes many other options and settings that you can explore as needed. For example, you can drag and drop and resize columns and rows. Many options are available when you right-click on a tab, column, row, or cell.

Figure 24-5.
A—A parts list references the content of the assembly model bill of materials, which extracts information from the assembly and each component. B—Basic options available in the Bill of Materials dialog box.

A

Option	Description
Model Data	Displays all components in the model, including components set as **Phantom** or **Reference**, which do not appear in a parts list.
Structured	When enabled, displays only components that will be extracted to a parts list; treats a subassembly as a subassembly in the parts list.
Parts Only	When enabled, displays only components that will be extracted to a parts list; treats each part in a subassembly as a part in the parts list.
300879-01 **Create Expression**	Displays and allows you to edit the contents of a cell. Changes are saved to the corresponding model and drawing.
Sort	Displays the **Sort** dialog box, from which you can modify the BOM order. Select the property to sort, such as **Item**, from the **Sort by** drop-down list, and specify ascending or descending order. If necessary, identify properties to sort second or third using the **Then by** areas.
Renumber	Displays the **Item Renumber** dialog box, from which you can renumber items according to a specified start value and increment.
Choose Columns	Displays the **Customization** dialog box. Drag and drop a property from the dialog box to a location in the BOM to add a column.
Add Custom iProperty Columns	Displays the **Add Custom iProperty Columns** dialog box, from which you can add a custom iProperty column when a column is unavailable from the **Customization** dialog box.

B

PROFESSIONAL TIP

Prepare the **BOM** before inserting a parts list to automate parts list definition. Use the **BOM** as a means of adjusting component properties without opening each component.

Parts List Tool

Access the **Parts List** tool and select a view to display the **Parts List** dialog box. See **Figure 24-6**. For applications in which the drawing does not include an assembly view, such as when the sheet displays only a parts list, choose a model using the **Select Document** drop-down list or **Browse** button. The **BOM Settings and Properties** area allows you to select display options and specify how components are extracted from the **BOM**. Pick an option from the **BOM View** drop-down list to choose the format to extract from the **BOM**.

The **Level** drop-down list is available when you pick the **Structured** format. Choose the **First Level** option to display parts and subassemblies, but not subassembly components. The **Min. Digits** drop-down list allows you to select the number of digits, from 1 to 6, to use for item numbering. Choose the **All Levels** option to display parts and subassemblies, with subassembly components nested under the subassembly. The **Delimiter** drop-down list allows you to specify the delimiter used to identify subassembly components.

The **Numbering** drop-down list is available when you pick the **Parts Only** format. Choose the **Numeric** option to identify parts with numbers, and use the **Min. Digits** drop-down list as needed. Choose the **Alpha** option to identify parts with letters, and use the **Case** drop-down list to select uppercase or lowercase letters. The **Structured (legacy)** and **Parts Only (legacy)** options allow you to apply a structured or parts only format as appropriate for previous versions of Inventor.

The **Table Wrapping** area includes options for wrapping, or dividing, a long parts list. Pick the **Enable Automatic Wrap** check box to wrap the parts list. Then choose the **Maximum Rows** radio button to split the parts list when the number of rows defined in the **Maximum Rows** text box is greater than the number of rows in the parts list. An alternative is to pick the **Number of Sections** radio button to split the parts list into the number of sections defined in the **Number of Sections** text box. Pick the **Left** or **Right**

Figure 24-6.
Using the **Parts List** tool and dialog box to add a parts list.
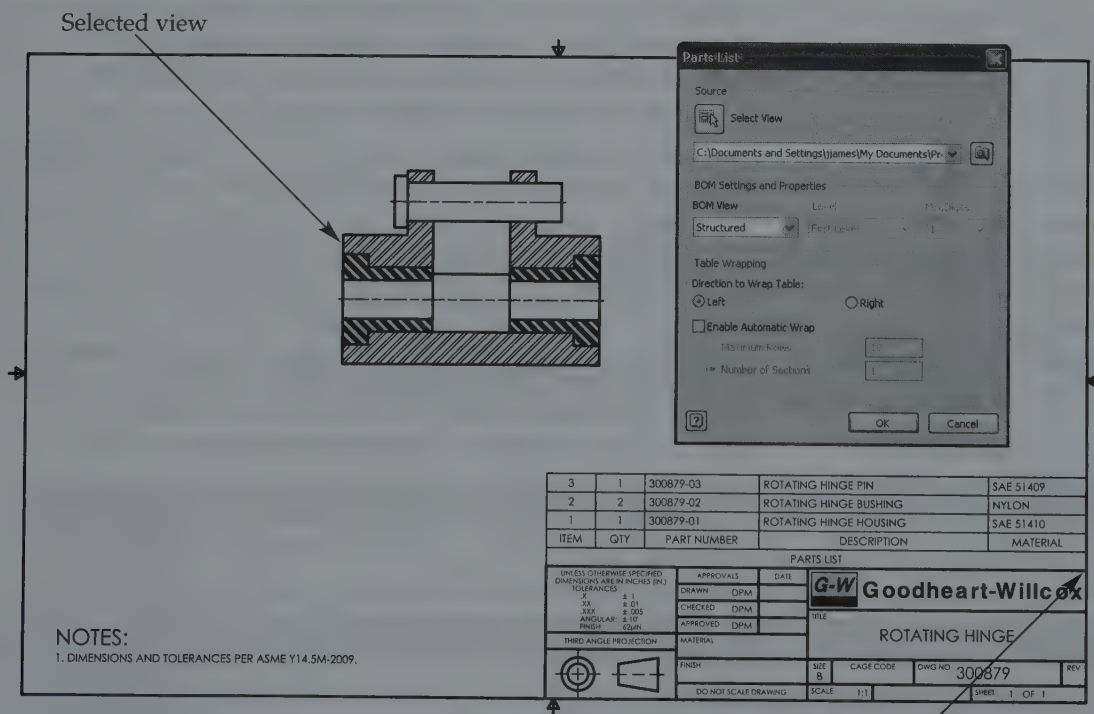
Figure 24-7.
An example of a parts list wrapped to create two sections.

8	1	15408—08	RACK PAD	SAE 4320						
7	1	15408—07	GEAR	SAE 4320	16	14	1		3/8—16UNC—2 X 1.00 HEX SOC CAP SCREW	STL
6	1	15408—06	HANDLE	SAE 1020	15	13	1	15408—13	COLUMN	SAE 1020
5	1	15408—05	SLEEVE	SAE 1020	14	12	1	15408—12	SCREW	SAE 1020
4	2	15408—04	BALL END	SAE 1020	13	11	1	15408—11	RACK	SAE 4320
3	1	15408—03	TABLE	SAE 1020					8—32UNC—2 X .50 HEX SOC CAP SCREW	
2	1	15408—02	TABLE PIN	SAE 1020	12	10	4			STL
1	1	15408—01	BASE	SAE 1020	11	9	1	15408—09	COVER PLATE	SAE 1020
ITEM	QTY.	DWG. NO.	DESCRIPTION	MATERIAL	2	ITEM	QTY.	DWG. NO.	DESCRIPTION	MATERIAL
			PARTS LIST		1				PARTS LIST	

radio button to specify the side to which the rows are wrapped. **Figure 24-7** shows an example of a parts list with two sections, wrapped to the right.

Pick the **OK** button to insert the parts list. A rectangle representing the parts list appears, attached to the cursor. The parts list should snap to a border, title block, or similar object to aid placement. Pick the location to generate the parts list.

Adjusting a Parts List

The **BOM** sets the default contents of parts list cells. The parts list style assigned to the active standard controls all other appearance characteristics, including the default columns. If the parts list does not appear correct, make changes to the parts list style assigned to the active standard and reinsert the parts list. You also have the option of adjusting the parts list for unique applications. Common examples of adjusting a parts list include dragging cell edges to change row and column spacing and re-sorting table contents when the model and drawing change. You can also override the contents of a cell, add or remove rows and columns, and export the parts list as a .txt or .csv file. Explore each option as needed to control the appearance of the parts list and data provided by the drawing.

The **Edit Parts List** dialog box, shown in **Figure 24-8A** and accessed by double-clicking on the parts list or right-clicking on the parts list and selecting **Edit Parts List...**, controls most editing options. **Figure 24-8B** describes basic options available in the **Edit Parts List** dialog box. Typing values in cells overrides the default properties specified in the **BOM**, as indicated by blue lettering and highlighting. A yellow background appears when the values you enter create discrepancies. Pick the **OK** button to accept the changes and exit the dialog box.

> **CAUTION**
>
> Avoid overriding default parts list values whenever possible. Save changes to the **BOM** when possible.

> **NOTE**
>
> The **Edit Parts List** dialog box includes many other options and settings that you can explore as needed. Many options are available when you right-click on a tab, column, row, or cell.

Exercise 24-4

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-4.

Figure 24-8.
A—The **Edit Parts List** dialog box. B—Basic options available in the **Edit Parts List** dialog box.

A

Option	Description
Column Chooser	Opens the **Parts List Column Chooser** dialog box, used to add columns to or remove columns from the parts list.
Group Settings	Opens the **Group Settings** dialog box to control, via the parts list, the display of separate assembly drawing components with the same properties.
Filter Settings	Displays the **Filter Settings** dialog box, from which you can filter to exclude all except specific items from appearing in the parts list.
Sort	Displays the **Sort** dialog box, from which you can modify the parts list order. Select the property to sort, such as **Item**, from the **Sort by** drop-down list, and specify ascending or descending order. If necessary, identify properties to sort second or third using the **Then by** areas.
Table Layout	Opens the **Parts List Table** dialog box, which allows you to override a variety of parts list layout characteristics.
Renumber Items	Overrides the sort option, value edits, or other numerical modifications, in order to renumber the parts list.
Save Item Overrides to BOM	Pick to save changes to the BOM for values that can be saved to the BOM.

B

Exercise 24-5

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-5.

Balloons

Balloons key the parts list to components in assembly views. The parts list and corresponding balloons usually reference the same data. When you change the order or value of item numbers in the BOM, both the parts list and the balloons change to reflect the edit. You can place balloons before or after you place the parts list. When you add balloons after a parts list, fewer settings are available because you have already established the link between the parts list and balloon data and the assembly model. You will notice the same effect when you place a parts list after balloons.

Balloon Tool

Access the **Balloon** tool to add balloons individually. Pick a component at the location where you want the leader to point. If the drawing already includes a parts list linked to the view, the **BOM Properties** dialog box does not appear and you are ready to place the first balloon. If you have not already added a parts list, the **BOM Properties** dialog box appears. The **Source** area contains the **File** display box, which identifies the location of the file associated with the selected drawing view and corresponding **BOM**. The **BOM Settings** area provides the same options available in the **BOM Settings and Properties** area of the **Parts List** dialog box. It allows you to specify how components are extracted from the **BOM** and to select display options. Pick the **OK** button to continue the process of adding balloons.

Pick a location for the balloon and then press [Enter] or right-click and select **Continue** to create the balloon. See **Figure 24-9**. Press [Esc], right-click and select **Done**, or access a different tool to exit. When placing additional balloons, use point alignment and angle snaps to help align and organize balloons in an easy-to-read format.

Ribbon
Annotate
> Table

Balloon

Auto Balloon Tool

Access the **Auto Balloon** tool to add all balloons to a view using the **Auto Balloon** dialog box. See **Figure 24-10**. Use the **Select View Set** button to pick a drawing view. Then select the **Add or Remove Components** button and select each component to receive a balloon. You can pick components in the graphics window or the browser. To deselect components, hold down [Ctrl]. The **Ignore Multiple Instances** check box is active by default, which does not allow you to attach balloons to multiple instances of components individually. For example, if an assembly contains three of the same bolts, only one bolt will receive a balloon if you check **Ignore Multiple Instances**.

Ribbon
Annotate
> Table
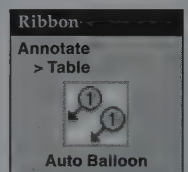
Auto Balloon

Figure 24-9.
Placing balloons using the **Balloon** tool. Notice the alignment and organization of balloons in an easy-to-read format.

1. Pick to select component and locate leader point

2. Pick to locate the balloon
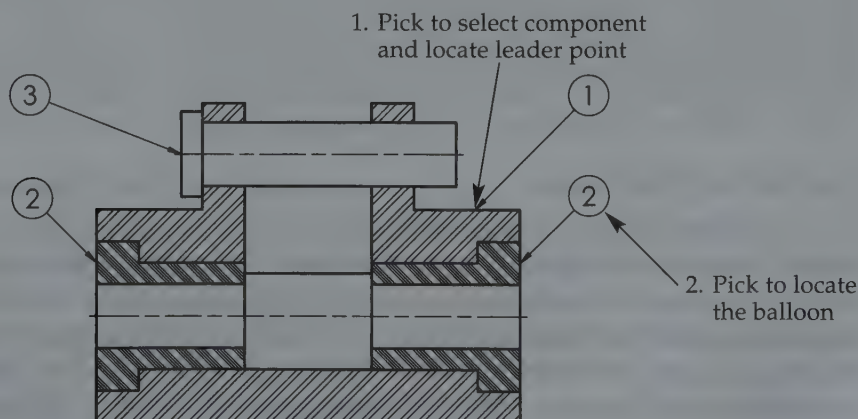
Figure 24-10.
Using the **Auto Balloon** tool and dialog box to place balloons. Notice how the balloon arrangement and location at which the leader points may not be ideal.

Around   Horizontal   Vertical
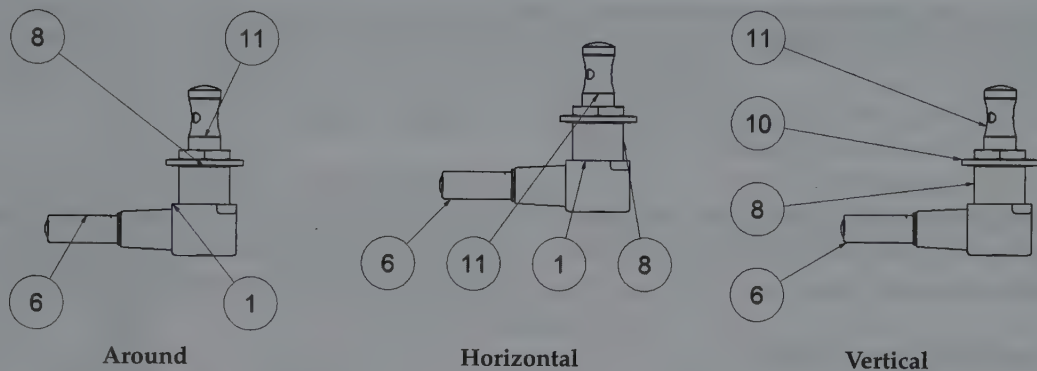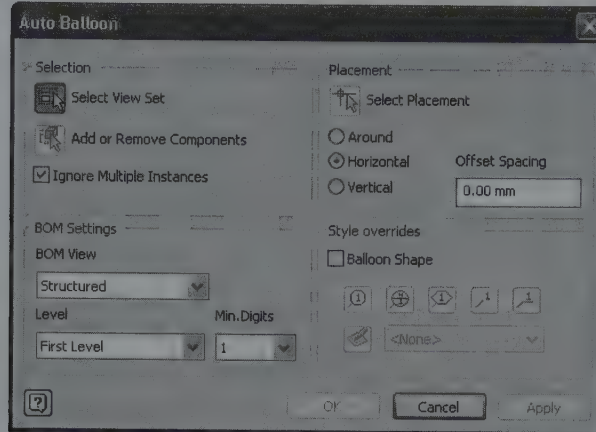
Next, define how the balloons are arranged around the assembly view using options in the **Placement** area. Pick the **Around** radio button to place balloons around the assembly in a rectangular configuration. Select the **Horizontal** radio button to place balloons horizontally aligned with each other at the distance specified in the **Offset Spacing** text box. Choose the **Vertical** radio button to place balloons vertically aligned with each other at the distance specified in the **Offset Spacing** text box. **Figure 24-10** shows examples of each placement option. Then use the **Select Placement** button to pick a location for the balloons.

The **BOM Settings** area includes the same settings available in the **BOM Properties** dialog box. The **Style Overrides** area allows you to override the default balloon display. Select the **Balloon Shape** check box and then pick a balloon type button to use a different balloon style. Pick the **User-Defined Symbol** button to choose a sketched symbol from the **Symbols** drop-down list. The available symbols correspond to drawing sketched symbols. Pick the **Apply** button to create balloons and remain in the tool. Pick the **OK** button to exit.

Adjusting Balloons

The **BOM** and parts list set the default contents of balloons, which is typically an item number corresponding to the parts list. The balloon style assigned to the active standard controls all other appearance characteristics. If the balloons are not correct, make changes to the balloon style assigned to the active standard. You can also adjust balloons for unique applications. Double-click or right-click on a balloon and select **Edit Balloon...** to access the **Edit Balloon** dialog box. See **Figure 24-11**.

Figure 24-11.
The **Edit Balloon**
dialog box.

Specify a new
balloon and parts list
number
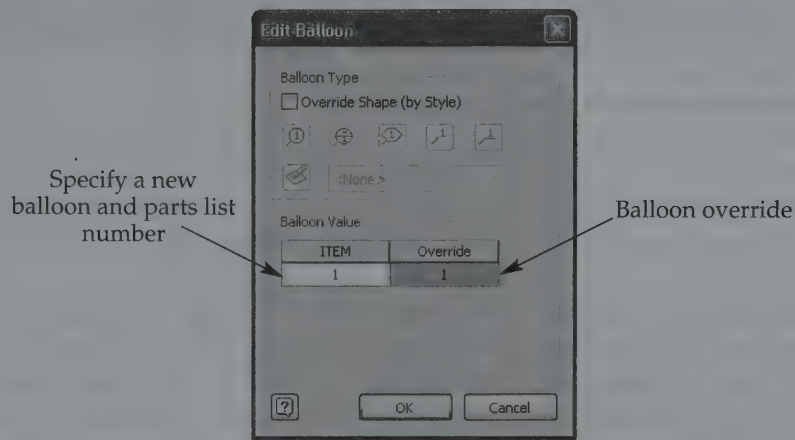
Balloon override

Deselect the **Override Shape (by style)** check box to override the default balloon settings. Pick a balloon type button to use a different balloon style. Pick the **User-Defined Symbol** button to choose a sketched symbol from the **Symbols** drop-down list. The available symbols correspond to drawing sketched symbols. You can override the current balloon value and the corresponding value in the parts list using the **Item** text box. Another option is to change the balloon value only, leaving the parts list number intact, using the **Override** text box. Pick the **OK** button to accept the changes and exit the dialog box.

CAUTION

Avoid overriding default balloon and parts list values whenever possible.

NOTE

You can relocate or delete a balloon or override a leader arrowhead using techniques similar to those for controlling other dimensions, such as leader notes.

Aligning Balloons

Adjust balloon alignment and organization to create an easy-to-read pattern when possible. Repositioning balloons is especially common when you use the **Auto Balloon** tool. Drag a balloon or the end of the leader to a new position as needed. You can use point alignment and angle snaps to arrange balloons while dragging. Another option is to select balloons to align and then right-click and pick an alignment option from the **Align** cascading submenu. The **Vertical** option aligns balloons vertically, and the **Horizontal** option aligns balloons horizontally. The **Vertical Offset** option aligns and offsets balloons vertically, and the **Horizontal Offset** option aligns and offsets balloons horizontally. The active standard controls the offset distance. Alignment options are similar to those available for the **Auto Balloon** tool, as shown in **Figure 24-10**.

Grouping Balloons and Adding Leaders

You can group balloons for closely related clusters of components, such as a bolt, washer, and nut. See **Figure 24-12**. Grouped balloons share the same leader, which typically connects to the most apparent or only visible component. To group balloons, right-click on a balloon that will maintain the current leader and position, and then pick **Attach Balloon**. Then select an additional component from the browser or graphics window. When you see the new balloon, pick the side of the existing balloon on which

Figure 24-12.
These grouped
balloons identify the
bolt, nut, and washer
in this view.
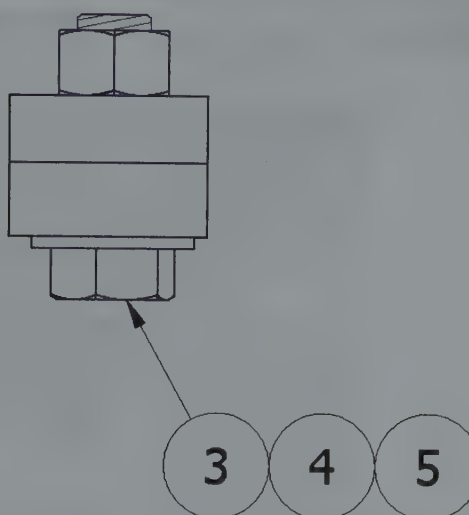
you want the new balloon to occur. To remove an attached balloon, right-click on the balloon to delete and select **Remove Balloon**.

Usually, one leader attaches to a balloon or group of balloons. However, you can add leaders by right-clicking on a balloon or balloon leader and selecting **Add Vertex/Leader**. Then pick the start point of the new leader and select a point on the existing balloon to connect the leader. See **Figure 24-13**. To remove a leader, right-click on the balloon, select **Delete Leader**, and then pick the leader to remove.

Exercise 24-6

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-6.

Exercise 24-7

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-7.

Figure 24-13.
Adding a leader to a
balloon.
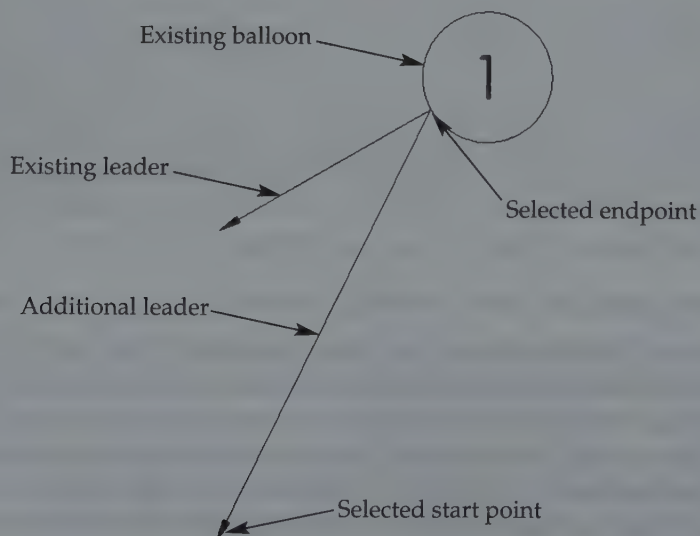
Weldment Drawings

You can add weld symbols to an assembly drawing to create a weldment drawing. However, for most applications, you should begin the process of developing a weldment drawing by creating a base view from a weldment model. Then add views, a parts list, and balloons as you would when preparing an assembly drawing. Weldment drawings also include welding symbols and other weld-specific annotations. See **Figure 24-14**.

To create the initial weldment view, access the **Base View** tool and select a weldment model (*.iam file) using options in the **Component** tab of the **Drawing View** dialog box. The **Representation** area functions the same as for adding an assembly base view, except the **Position** list box is unavailable because weldment components are fixed. As shown in **Figure 24-15**, the **Model State** tab provides a **Weldment** area in addition to the **Reference Data** area available for placing an assembly base view. Select the radio button corresponding to the weldment development stage you want to represent in the view. The **Components** drop-down list becomes enabled when you select the **Preparations** radio button. Pick the **Assembly** option to display the entire prepared assembly, or choose individual components to show specific prepared components.

The **Display Options** tab contains **Model Welding Symbols** and **Weld Annotations** check boxes specific to creating a weldment drawing. Pick the **Model Welding Symbols** check box to display welding symbols from the model in the view. Select the **Weld Annotations** check box to include items such as bead representations, caterpillars, and end fills. The **Orientation**, **Scale**, **Label**, and **Style** areas function the same as for

---

Figure 24-14.
An example of a dimensioned weldment drawing with welding symbols and weld annotations extracted from the model.
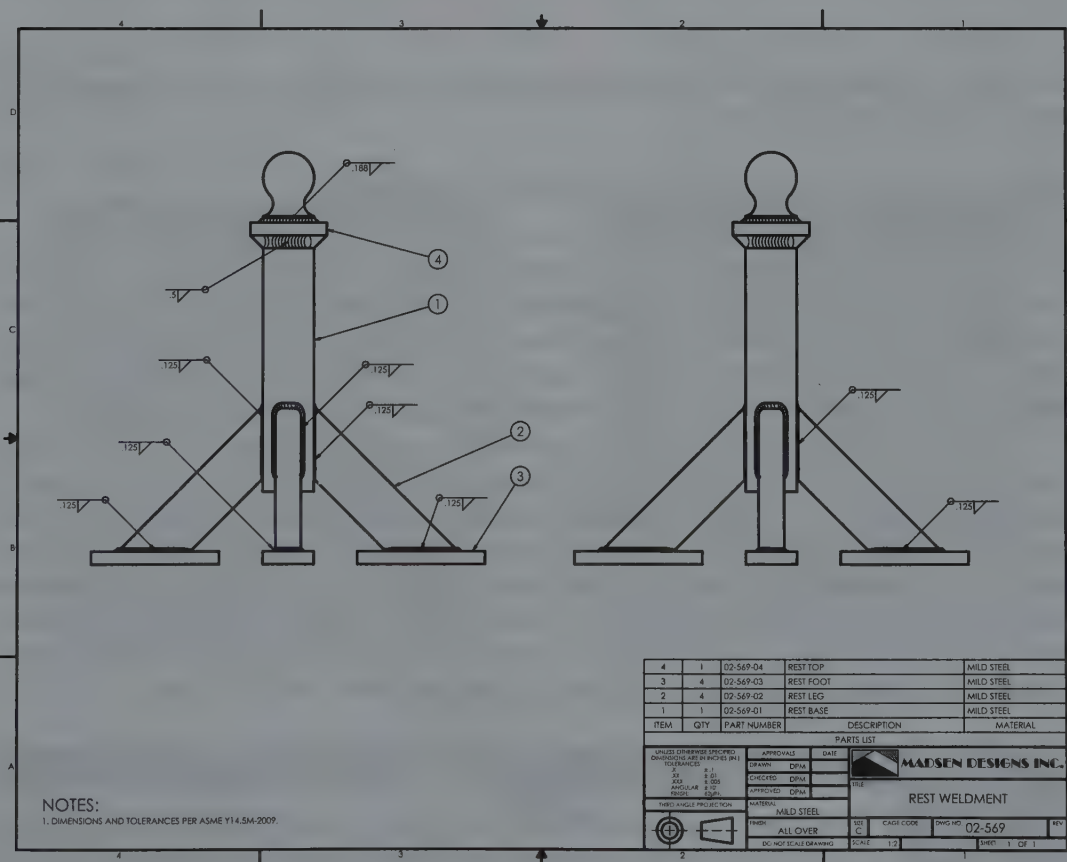
Figure 24-15.
Using the **Base View** tool to create a weldment view by referencing a weldment model.

Pick to locate

Weldment-specific options
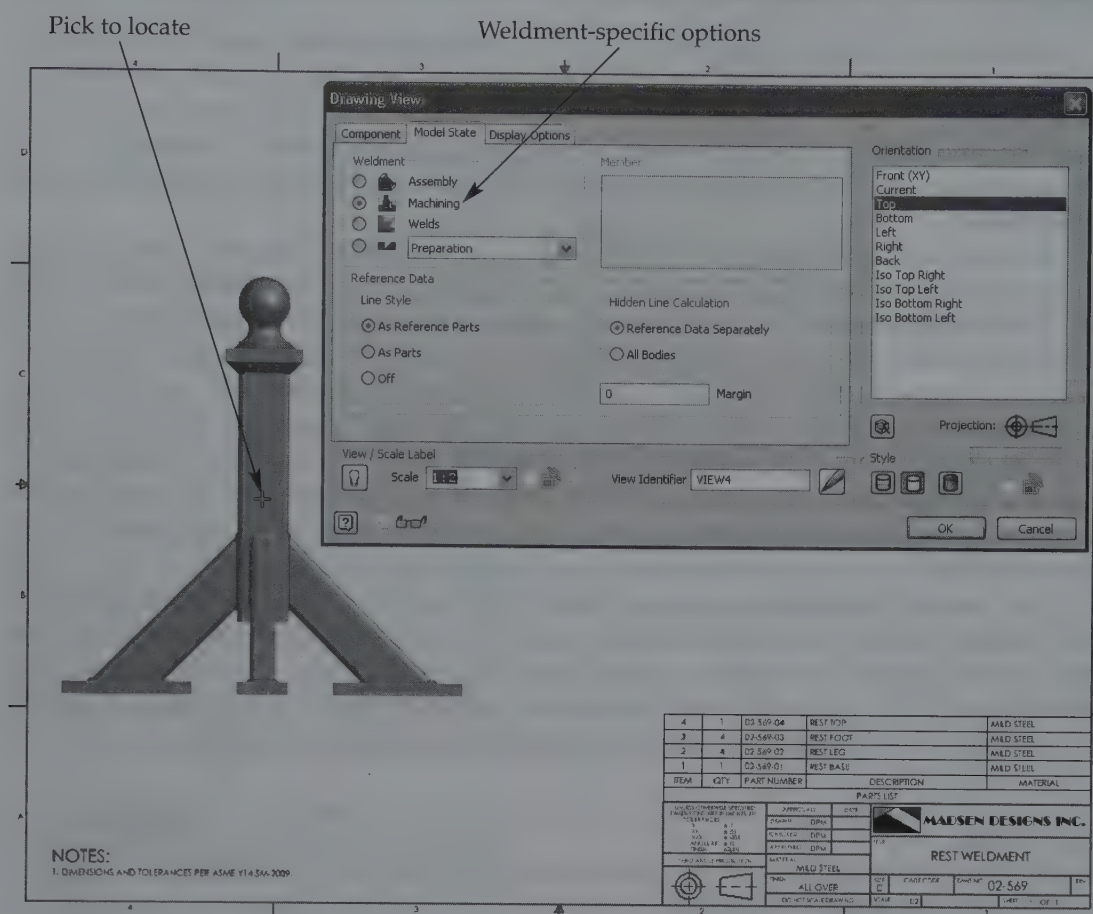
creating an assembly base view. To place the base view, pick a location in the graphics window or pick the **OK** button.

Model Welding Symbols and Weld Annotations

Welding symbols and weld annotations often occur in a weldment model. You can extract model welding symbols and weld annotations that are planar to a drawing view by selecting the **Model Welding Symbols** and **Weld Annotations** check boxes from the **Display Options** tab of the **Drawing View** dialog box. You can also right-click on a view in the browser or graphics window and select the **Get Welding Symbols** and **Get Weld Annotations** options from the **Get Model Annotations** cascading submenu.

To make changes to model welding symbols or weld annotations, edit the model or adjust the active standard. Use model dimension editing operations to adjust or hide model welding symbols or weld annotations. To hide all model welding symbols or weld annotations, right-click a view in the browser or graphics window and deselecting **Model Welding Symbols** or **Model Annotations** from the **Annotation Visibility** cascading submenu. You can make basic changes to a model caterpillar by double-clicking or right-clicking on a weld bead representation and selecting **Edit Caterpillar...** to display the **Weld Caterpillars** dialog box, explained later in this chapter.

> **NOTE**
>
> You cannot adjust welding symbols or weld annotations in the drawing environment to make changes to welds in the model.

Exercise 24-8

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-8.

End Fills

Use the **End Fill** tool to add end fills if model annotations are unavailable. The **Type** tab of the **End Fill** dialog box is shown in **Figure 24-16A**. Select an end fill shape button from the **Preset Shape** area, and then use the options in the **Size** area to specify the size of the end fill. An alternative is to sketch a region to represent the end fill by picking the **Custom Shape** button in the **Custom Shape** area.

Pick the **Options** tab, shown in **Figure 24-16B**, and use the settings in the **Fill** area to define the end fill display characteristics. Check **Solid Fill** to add a solid fill to the region, or use the **Hatch** drop-down list and **Scale** text box to specify the appearance of a hatch pattern. The **Color** button allows you to override the default color using the **Color** dialog box. To create the end fill, pick one or two points on drawing view objects, depending on the shape, and then select the side on which to add the end fill. See **Figure 24-16C**. Pick the **Apply** button to create the end fill and remain in the tool. Pick the **OK** button to exit.

> **Ribbon**
> Annotate
> > Symbols
>
> **End Fill**

Caterpillars

> caterpillar: 2D representation of a weld bead perpendicular to the line of sight.

Use the drawing **Caterpillar** tool to add *caterpillars* if model annotations are unavailable. The **Style** tab of the **Weld Caterpillars** dialog box is shown in **Figure 24-17A**. Use the **Edges** button to pick edges in the drawing to place the caterpillar. The **Start/Stop** button allows you to select points where the weld begins and ends. Use the **Direction**

> **Ribbon**
> Annotate
> > Symbols
>
> **Caterpillar**

Figure 24-16.
A—The **Type** tab of the **End Fill** dialog box. B—The **Options** tab of the **End Fill** dialog box. C—Creating a V-type end fill.

A   B

C

1. Select a point   2. Pick the end fill side
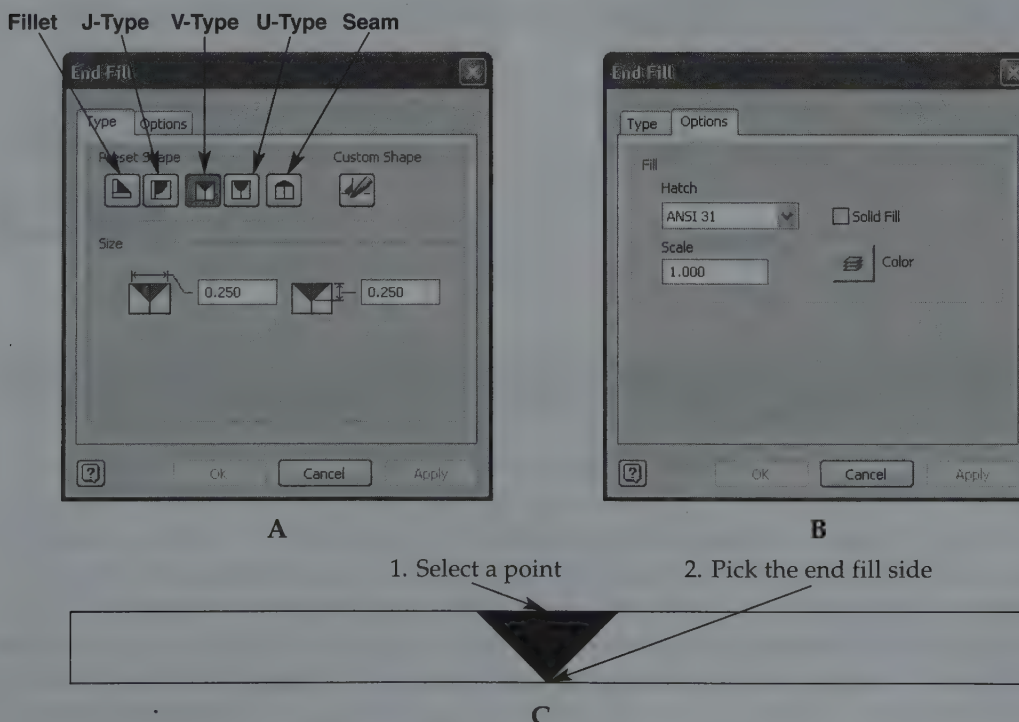
button to redefine the weld flow direction. Pick the **Partial** button to place a caterpillar next to an edge, such as when representing a fillet weld. Select the **Full** button to place a caterpillar along both sides of an edge, such as when representing a V- or square-groove weld. Check **Stitch** to define an intermittent weld.

Pick the **Options** tab, shown in **Figure 24-17B**, and use the text boxes and flyout in the **Legs** area to specify the width, angle, arc percentage, spacing, and line weight of the caterpillar. The **Stitch Options** area is available if you check **Stitch** in the **Style** tab, and allows you to create an intermittent weld bead by specifying the length of each weld bead and the center-to-center distance between welds. Check **Seam Visibility** to display the edge hidden by the caterpillar. See **Figure 24-17C**. Pick the **Apply** button to create the caterpillar and remain in the tool. Pick the **OK** button to exit.

Welding Symbols

Use the **Welding Symbol** tool to add welding symbols if model welding symbols are unavailable. Create a welding symbol in a drawing using the same process you use to create a welding symbol in a model. See **Figure 24-18**. The only difference between the **Welding Symbol** dialog box in the model and drawing environments is that the **Symbol** area becomes available when you add a welding symbol to a drawing. Pick the **Add** button to insert an additional reference line. Use the **Next** and **Previous** buttons

Figure 24-17.
A—The **Style** tab of the **Weld Caterpillars** dialog box. B— The **Options** tab of the **Weld Caterpillars** dialog box. C— Creating a caterpillar.
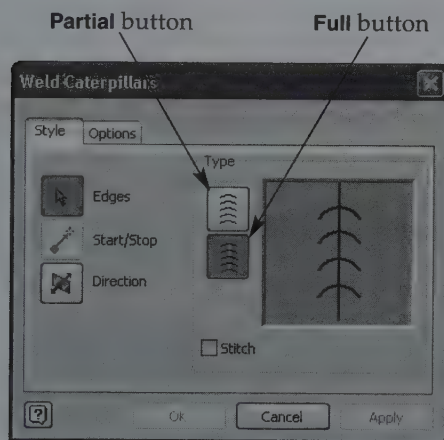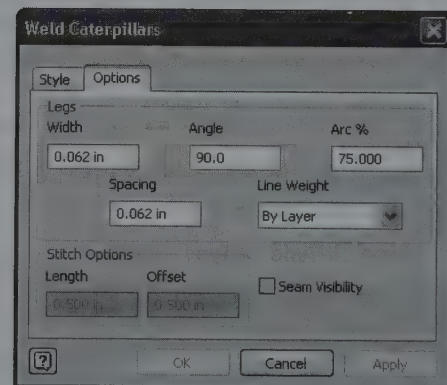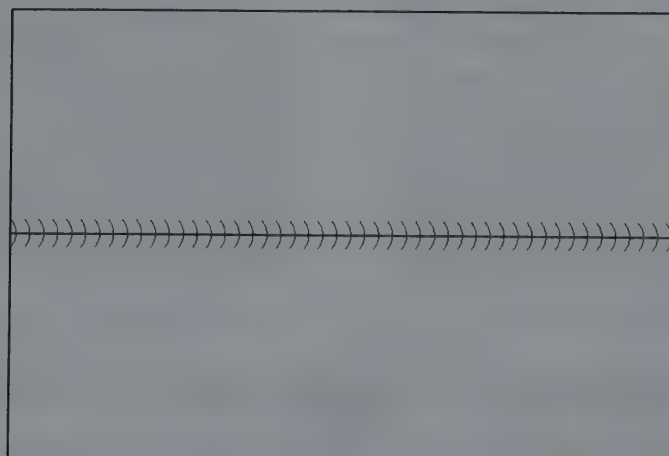
Figure 24-18.
Placing a welding symbol using the **Welding Symbol** tool and dialog box.

1. Select an object or a point

2. Pick the symbol location

.375x.5 2-4 SMAW
.25x.375 4-6
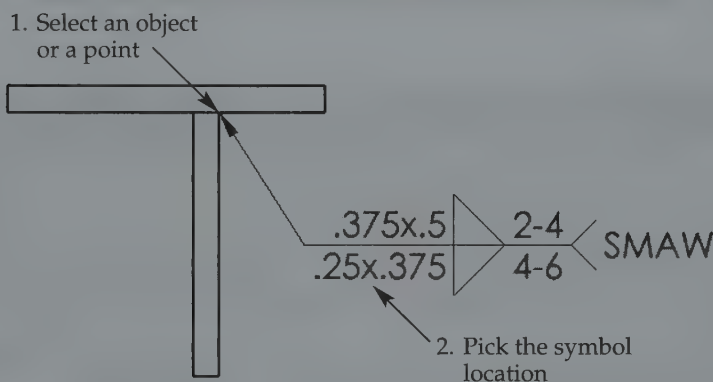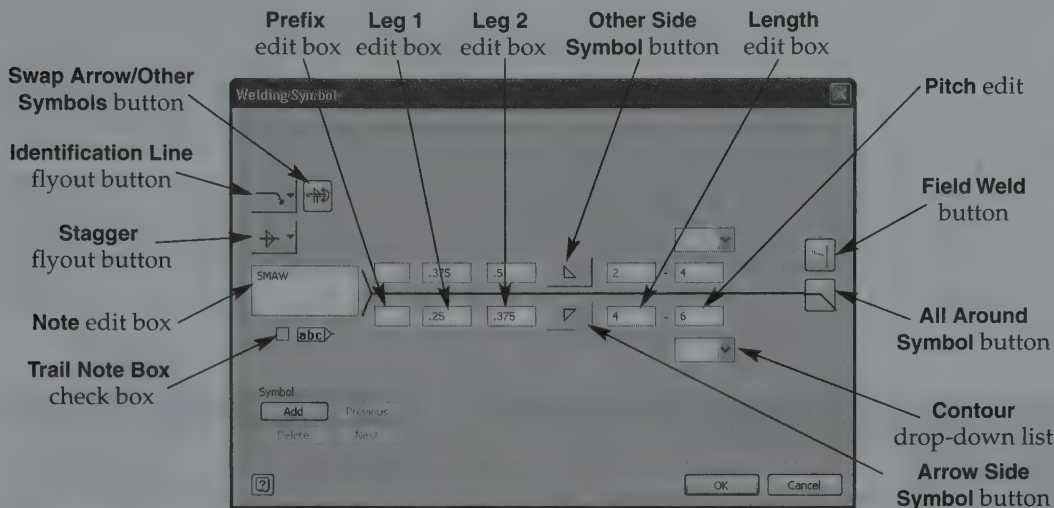
to navigate to different reference lines to add content. The **Delete** button allows you to remove a reference line. Pick the **OK** button to create the weld symbol. Press [Esc], right-click and select **Done**, or access a different tool to exit.

Exercise 24-9

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-9.

Multiple Sheets

A set of working drawings generally includes an assembly or weldment drawing and detail drawings. You can use multiple sheets in a drawing file to store each document, similar to the process of storing prints or plots in a folder. This technique eliminates the need to create several drawing files for each drawing. Several options are available for developing a multiple-sheet drawing file. A common approach to starting a new set of drawings is to create the assembly or weldment drawing and then use blank sheets or sheet formats to add detail drawings. You can also insert existing sheets if available.

Adding a Sheet

Access the **New Sheet** tool to insert a new sheet based on the characteristics of the active sheet. The new sheet is active and appears at the bottom of the list of sheets in the browser. See **Figure 24-19**. All parameters, including the name, size, and drawing resource content, such as a border and title block, are copied from the active sheet to the new sheet. You must recreate or copy and paste items not available from the **Drawing Resources** folder, such as general notes.

If the drawing includes multiple sheets, activate the sheet to reuse when adding the new sheet by double-clicking on the sheet in the browser or right-clicking on the sheet and selecting **Activate**. If you do not want to use the same border and title block, use the **Undo** tool until the border and title block are removed. Access the **Edit Sheet** dialog box to make additional changes to the sheet.

sheet format: A multi-element sheet template stored in the drawing file.

> **NOTE**
>
> A *sheet format* allows you to create a new sheet with common sheet items such as border and title block, but also predefined views. Chapter 22 describes creating and using sheet formats.

Copying and Pasting a Sheet

Copying and pasting a sheet from a separate file brings the sheet with all information, including drawing views and annotations, into the new file. Copying and pasting sheets is common for converting individual drawing files to a single multi-sheet drawing and for reusing standard drawings. Open the drawing file with the sheet to copy, right-click on the sheet in the browser or graphics window, and select **Copy** or pick the sheet and press [Ctrl]+[C]. Next, with the source file still open, access the drawing file into which you want to place the copy, right-click on the graphics window or the file name in the browser, and select **Paste** or press [Ctrl]+[V].

Working with Multiple Sheets

Figure 24-19 shows the browser display of a drawing file with five drawing sheets. The active sheet appears in the graphics window. Only one sheet can be active at a time. The order of sheets in the browser determines sheet number or page number. You can drag a sheet above or below another sheet to change the order. Sheet order also defines title block information when you are using drawing properties. For example, if the drawing includes five sheets, and each sheet has a title block with the **Sheet Number**

Figure 24-19.
An example of the browser display for a multiple-sheet drawing.

Active sheet
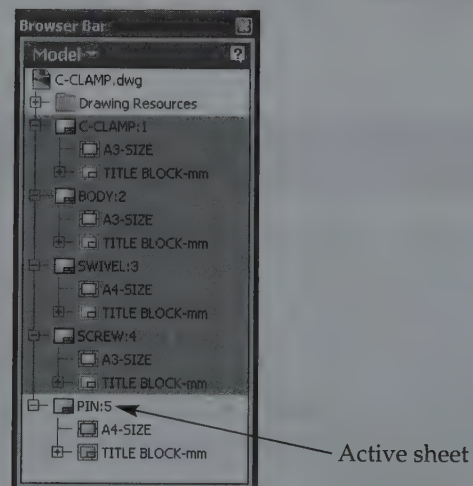
and **Number of Sheets** property fields, the first sheet in the browser is sheet 1 of 5, and the last sheet is sheet 5 of 5.

Although you cannot copy a sheet within the same drawing file, in some situations you may want to copy or move a view from one sheet to another. To copy a view, right-click on the view and select **Copy** or pick the view and press [Ctrl]+[C]. Right-click on the sheet on which you want to place the view and select **Paste** or press [Ctrl]+[V]. To move a view to a different sheet, drag and drop the view using the browser. A shortcut forms when you move a base or dependent view. The shortcut indicates that there is a relationship between two views in different sheets. To view the relationship, right-click on the shortcut icon in the browser and select **Go To...** to open the corresponding sheet.

Exercise 24-10

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-10.

Engineering Changes

Design changes are common throughout the life of a product. A *revision history block* and *revision symbols* allow you to document changes made to a design after release to manufacturing. See **Figure 24-20**. This textbook focuses on the following process to apply and document engineering changes:

revision history block (revision table, revision block): A table that records drawing changes, usually placed in the upper-right corner of the drawing.

revision symbol (revision tag): A symbol that identifies the location at which an engineering change occurs. Revision symbols correspond to entries in the revision table
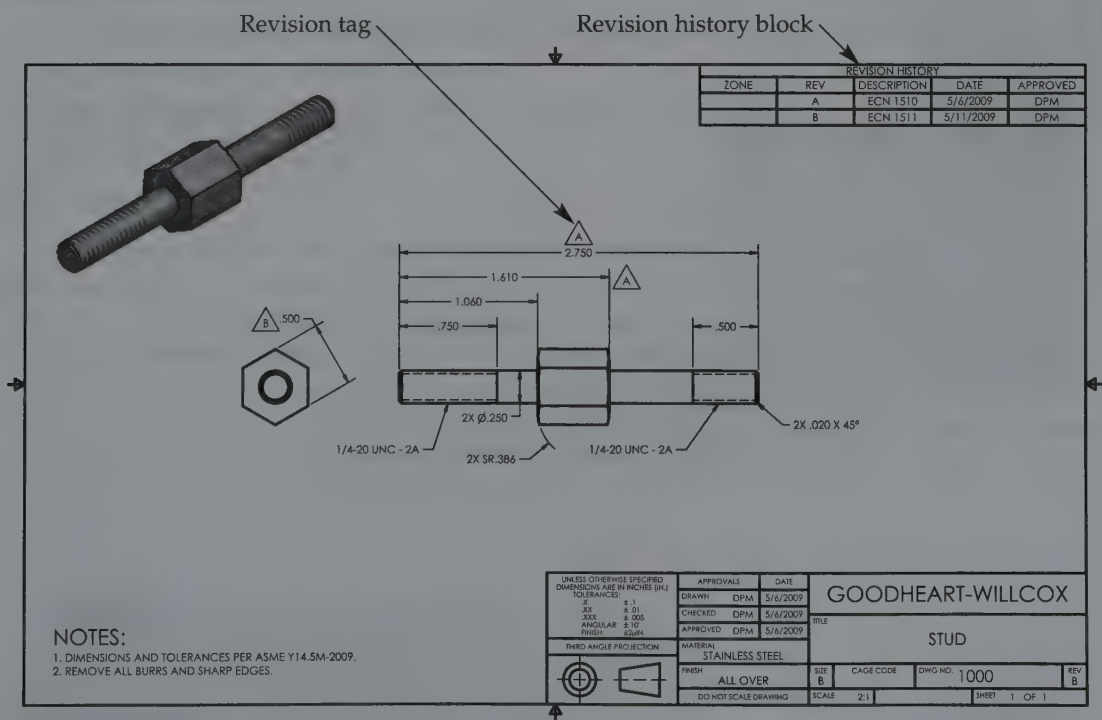
Figure 24-20.
A part drawing with a revision history table and corresponding revision tags documenting engineering changes.

First Revision:

1. Edit the model and all associated models according to the *engineering change request (ECR)*. Change the revision number iProperty in the modified model from 0 to A. This will update the revision number on the drawing title block.
2. Add a revision history block to the affected sheet in the drawing.
3. Place revision tags that correspond to the first revision.

Second Revision:

1. Edit the model and all associated models according to the ECR. Change the revision number iProperty in the modified model from A to B.
2. Add a row to the revision history block that corresponds to the second revision.
3. Place revision tags that correspond to the second revision.

Continue to apply the revision process as design changes occur.

Revision Table

Access the **Revision Table** tool to create a revision history block using the **Revision Table** dialog box. See **Figure 24-21**. Pick the **Entire Drawing** radio button to duplicate the revision history on all sheets in the drawing file. All revision history blocks in the drawing that are created using the **Entire Drawing** option display the same information. You cannot add a unique revision to a specific sheet. The **Entire Drawing** option is appropriate only for applications in which a set of working drawings requires a global change. This textbook focuses on using the **Active Sheet** option to apply revisions that are independent of the drawing file and other sheets. The **Active Sheet** option provides more control over the content of the revision block per sheet and documents revisions on a sheet-by-sheet basis.

Check **Auto-Index** to allow Inventor to assign a revision letter or number. Pick the **Alpha** radio button to apply the common system of letters, or pick the **Numeric** radio button to use numbers. Use the **Start Value** text box to specify the first revision letter or number. If you deselect the **Auto-Index** check box, you must edit the revision table to add a revision letter or number. Check **Update Property on Revision Number Edit** to

Figure 24-21.
Creating a revision history block using the **Revision Table** tool and dialog box.
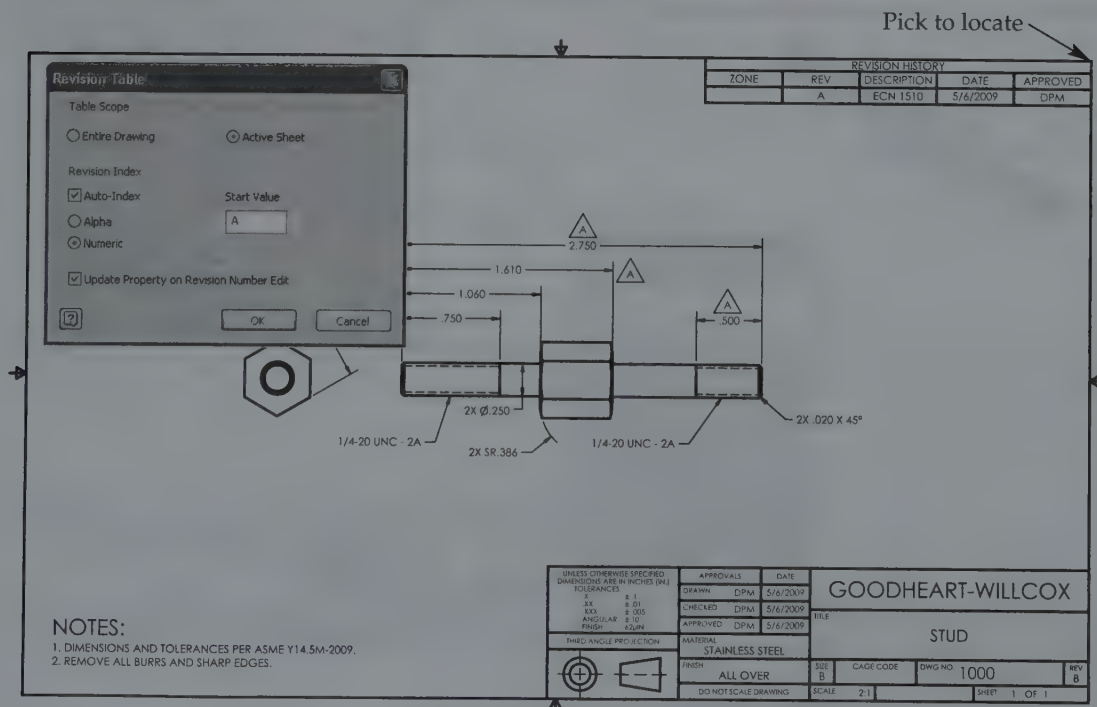
add the appropriate revision letter or number to the **Revision** text box of the **Edit Sheet** dialog box.

Pick the **OK** button to insert the revision table. A rectangle representing the revision table extents appears, attached to the cursor. The revision table should snap to a border, which is most common, but you can also snap it to a title block or similar object to aid placement. Pick the location to generate the revision table. To add a row to the revision table, right-click on the revision table in the graphics window or browser and select **Add Row**. Use the **Delete Row** option to remove a row.

> **NOTE**
>
> A single sheet can include both an entire drawing revision block and an active sheet revision block, but this practice is uncommon.

Adjusting a Revision Table

The **Revision Table** dialog box and iProperties set the default contents of a revision table. The revision table style assigned to the active standard controls all other appearance characteristics, including the default columns. If the revision table is not correct, make changes to the Revision Table style assigned to the active standard. You also have the option of adjusting the revision table for unique applications. For example, the revision table shown in **Figure 24-21** shows an override applied to the description value to reference a sequential *engineering change notice (ECN)* number. If necessary, you can drag the revision table to reposition it, or drag cell edges to change row and column spacing.

Most editing options are controlled in the **Revision Table: Drawing Scope** dialog box, shown in **Figure 24-22A**, which you can access by double-clicking on the revision table or right-clicking on the revision table and selecting **Edit**. **Figure 24-22B** describes basic options available in the **Revision Table: Drawing Scope** dialog box. Typing values in cells overrides default iProperties, as indicated by blue lettering and highlighting. Pick the **OK** button to accept the changes and exit the dialog box.

engineering change notice (ECN): A document that records changes made during a revision.

> **NOTE**
>
> The **Revision Table: Drawing Scope** dialog box includes other options and settings that you can explore as needed. Many options are available when you right-click on a column, row, or cell.

Revision Tags

Revision tags key the revision table to revised items, such as dimensions, notes, or title block information on a sheet. Refer again to **Figure 24-20**. You typically add revision tags immediately after creating a revision table and each time you add a revision. Access the **Revision Tag** tool to place revision tags. Select a position in space, or pick a point or feature to begin placement. To locate the tag at the selection, press [Enter] or right-click and select **Continue**. To connect the tag to a leader, pick a second point and press [Enter] or right-click and select **Continue**. Place additional copies of the same revision tag, if required, or press [Esc], right-click and select **Done**, or access a different tool to exit.

Ribbon
Annotate
> Table
> Revision
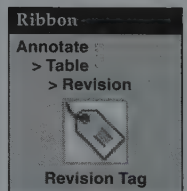
Revision Tag

Exercise 24-11

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-11.

Figure 24-22.
The **Revision Table: Drawing Scope** dialog box and basic options available for editing a revision tab.

A

Option	Description
Column Chooser	Opens the **Revision Table Column Chooser** dialog box, used to add columns to or remove columns from the revision table.
Sort	Displays the **Sort Revision Table** dialog box, where you can modify the revision table order. Select the property to sort, such as **Item**, from the **Sort by** drop-down list, and specify ascending or descending order. If necessary, identify properties to sort second or third using the **Then by** areas.
Table Layout	Opens the **Revision Table Layout** dialog box, used to override a variety of revision table layout characteristics.
Insert Row	Inserts a blank row below the active row.
Remove Row	Removes a row or revision.
iProperties	Displays the **iProperties** dialog box associated with the drawing file. By default, some revision block content originates from drawing iProperties.
Add Revision Row(s)	Adds a row or revision.
Update Property to Revision Number	Adds the appropriate revision letter or number to the **Revision** text box on the **Edit Sheet** dialog box.

B

Printing Drawings

To print or plot a drawing, first access the **Print Setup** tool to preset print settings using the **Print Setup** dialog box. See **Figure 24-23**. Use the **Name** drop-down list to select a plot device and, if necessary, pick the **Properties** button to adjust plot device settings using the **Properties** dialog box. The plot device specifications appear in the **Printer** area. Select the sheet size from the **Size** drop-down list and choose where the paper comes from using the **Source** drop-down list. Select the **Portrait** radio to rotate and print the document in a portrait, or vertical, orientation, or choose the **Landscape** radio button to rotate and print the document in a landscape, or horizontal, orientation. Pick the **Network** button to access the **Connect to Printer** dialog box, in which you can connect to a shared network printer and plot to that device. Pick the **OK** button to accept the changes and exit print setup.

Access the **Print Preview** tool to preview the print using the current print specifications. See **Figure 24-24**. Preview other pages to print with the current print job using the **Next Page**, **Prev Page**, or **Two Page** buttons. Zoom in or out to review the print preview using the **Zoom In** or **Zoom Out** buttons. Pick the **Print...** button to initiate the print or select the **Close** button to terminate the preview.

Figure 24-23.
The **Print Setup** dialog box.
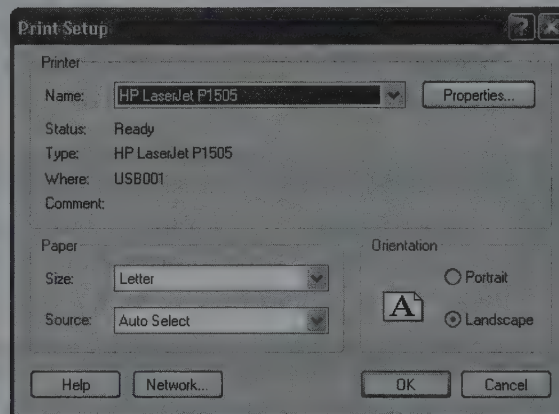
Figure 24-24.
A preview of a print job.

Application Menu

Print

Print

Quick Access

Print

Type

[Ctrl]+[P]
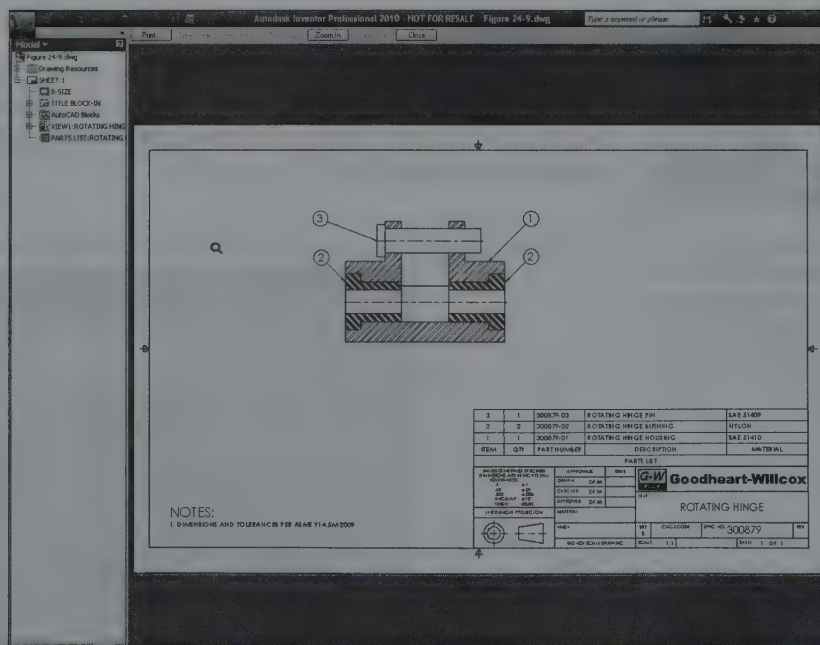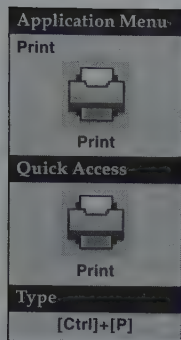
Access the **Print** tool to prepare, preview, and print a drawing using the **Print Drawing** dialog box. See **Figure 24-25**. Use the **Name** drop-down list to select a plot device and, if necessary, pick the **Properties** button to adjust plot device settings using the **Properties** dialog box. The **Print Range** area defines which sheets in the drawing file print. Select the **Current Sheet** radio button to print only the active drawing sheet, or select the **All Sheets** radio button to print all of the sheets in the drawing file. To print a range of sheets, pick the **Sheets in Range** radio button. Then enter the sheet to print first in the **From** text box and the sheet to print last in the **To** text box. Select the **Print excluded sheets** check box to override the **Exclude From Print** option specified in the **Edit Sheet** dialog box and print the sheets specified as excluded.

The **Settings** area provides additional controls. Specify the number of copies to print using the **Number of Copies** text box. Check **Rotate By 90 Degrees** to rotate the print 90° on the sheet. Select the **All Colors As Black** check box to print all objects black. Pick the **Remove Object Line Weights** check box to print all geometry using the same line weight.

Use options in the **Scale** area to set the required print scale. Select the **Model 1:1** radio button to print the sheet at full size, correctly representing view scale. The **Model 1:1** option is appropriate for most applications. The **Tiling Enabled** check box tilts, or arranges, sheets for printing on multiple pages. Select the **Best Fit** radio button to scale the sheet according to the paper size when scale is not important. To apply a scale factor other than 1:1, pick the **Custom** radio button and specify the scale in the **Custom** text box. For example, enter 25.4 to print an inch drawing on a metric sheet. Choose the **Current Window** radio button to print the display of the current window without a specified scale factor.

Select the **Preview...** button to display the **Print Preview** dialog box and preview the intended print job. Pick the **OK** button to print.

PROFESSIONAL TIP

Previewing a print is good practice, especially for large drawings that take a long time to print and use a large sheet of paper.

NOTE

You can print or export a drawing as a PDF of DWF file, which is common for displaying drawings electronically.

Exercise 24-12

Access the Student Web site (www.g-wlearning.com/CAD) and complete Exercise 24-12.

Figure 24-25.
The **Print Drawing** dialog box for printing a drawing.
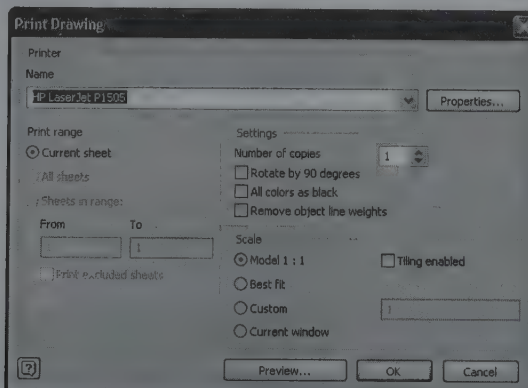
Chapter Test

1. What are working drawings?
2. Briefly describe an assembly drawing.
3. Describe an exploded assembly drawing.
4. What is reference data?
5. Briefly describe a general assembly drawing.
6. Define *detail assembly drawing*.
7. What is an erection assembly drawing?
8. Describe a pictorial assembly drawing.
9. Define *alternate position drawing view*.
10. Identify the function of a parts list.
11. What is a BOM?
12. What are balloons?
13. Describe a revision history block.
14. Explain the function of an engineering change request and give the proper abbreviation.
15. Describe an engineering change notice.

Problems

Instructions:

- *Begin a new drawing file for each of the projects.*
- *Create a complete set of working drawings using each of the specified files.*
- *The first sheet should be an assembly or weldment with balloons and a parts list.*
- *Create additional sheets for each detail.*
- *Use an appropriate sheet size border and title block for each drawing sheet.*
- *Create necessary drawing views at an appropriate scale, and fully dimension and annotate each drawing as required.*
- *Define the drawing properties in the drawing and individual model **iProperties** dialog boxes so that properties display correctly in the title block.*

1. **Title:** PUSH PIN
   Units: Inch
   Template: Drawing-IN.idw
   Project: PUSH PIN
   Assembly File: P17-3.iam
   Save as: P24-1.idw
   Specific Instructions: Create a general assembly drawing and detail drawings for each part.

2. **Title:** FORK
   Units: Metric
   Template: Drawing-mm.idw
   Project: FORK
   Assembly File: P18-2.iam
   Save as: P24-2.idw
   Specific Instructions: Create a general assembly drawing and detail drawings for each part.

3. **Title:** STUDDED HUB
   Units: Inch
   Template: Drawing-IN.idw
   Assembly (weldment) File: P20-2.iam
   Save as: P24-3.idw
   Specific Instructions: Create a general weldment assembly drawing and detail drawings for each part.

4. **Title:** BOTTLE ASSEMBLY
   Units: Inch or Metric
   Template: Drawing-IN.idw or Drawing-mm.idw
   Project: BOTTLE
   Presentation File: P21-2.ipn
   Save as: P24-4.idw
   Specific Instructions: Create a general assembly drawing and detail drawings for each part.

5. **Title:** WATCH BAND CLASP
   Units: Inch or Metric
   Template: Drawing-IN.idw or Drawing-mm.idw
   Project: WRIST WATCH
   Assembly File: P17-6.iam
   Save as: P24-5.idw
   Specific Instructions: Create a general assembly drawing, a subassembly drawing, and detail drawings for each part.

6. **Title:** ROTATING HINGE ASSEMBLY
   Units: Inch or Metric
   Template: Drawing-IN.idw or Drawing-mm.idw
   Project: ROTATING HINGE ASSEMBLY
   Assembly File: P18-4.iam
   Save as: P24-6.idw
   Specific Instructions: Create a pictorial assembly drawing and detail drawings for each part.

7. **Title:** HOUSING ASSEMBLY
   Units: Inch
   Template: Drawing-IN.idw
   Project: HOUSING
   Assembly (weldment) File: P20-1.iam
   Save as: P24-7.idw
   Specific Instructions: Create a general weldment drawing and detail drawings for each part.

8. **Title:** STAND
   Units: Inch or Metric
   Template: Drawing-IN.idw or Drawing-mm.idw
   Project: STAND
   Assembly (weldment) File: P20-4.iam
   Save as: P24-8.idw
   Specific Instructions: Create a general weldment drawing and a multidetail drawing documenting each part.

9. **Title:** SHOCK ABSORB SYSTEM
   Units: Inch or Metric
   Template: Drawing-IN.idw or Drawing-mm.idw
   Project: UTILITY VEHICLE
   Presentation File: P21-8.ipn
   Save as: P24-9.idw
   Specific Instructions: Create a general assembly drawing, a subassembly drawing, and detail drawings for each part.

10. **Title:** MULTI-ADJUSTER SLIDE
    Units: Inch
    Templates: Part-IN.ipt (part models), Assembly-IN.iam (assembly models), and Drawing-IN.idw (drawings)
    Project: IONIC SYNTHESIZER
    Specific Instructions: Use the model, drawing, and parts list shown to create part, subassembly, assembly, and drawing files as needed to design and document a product similar to the product shown. Create a project to contain all files and use an appropriate file naming system.

Problems – Chapter 24

NOTES:
1. DIMENSIONS AND TOLERANCES PER ASME Y14.5M-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.
3. TORQUE TO 125 PSI DURING ASSEMBLY.

2X GTAW
.0625x.0875
2X GTAW
.0875x.125

ITEM	QTY	PART NUMBER	DESCRIPTION
14	1	KNOB-OG0956HY	COIL KNOB
13	1	KNOB-009867F	SIDE KNOB
12	1	BAM-CW-2635	COIL WASHER
11	1	BAM-CC-6712	COIL COVER
10	3	JRM-IB-583	INSIDE BUSHING
9	1	BAM-MC-8976	MAIN COIL
8	1	JRM-MB-567	MAIN BUSHING
7	3	JRM-NB-245	NARROW BUSHING
6	1	3/8-24 - 0.75	HEX WSHR SCR
5	5	DAM-MFG-1047	MAIN FRAME GRIP
4	5	1/4-20 - 0.5	BTN HD. HEX SCH, B18.3
3	1	DPM-BG-1024	BASE GASKET
2	1	DPM-BC-0023	BASE COVER
1.3	1	JRM-SB-914	SMALL BUSHING
1.2	1	JRM-B-846	BUSHING
1.1	1	DAM-F-2147	FRAME
1	1	DAM-FW-0047	FRAME WELDMENT
ITEM	QTY	PART NUMBER	DESCRIPTION

PARTS LIST

UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES (IN.)
TOLERANCES:
.X ± .1
.XX ± .01
.XXX ± .005
ANGULAR: ± 10°
FINISH: 62µIN

THIRD ANGLE PROJECTION

APPROVALS	DATE
DRAWN Dave Madsen	1/15/2003
CHECKED	
APPROVED	

MATERIAL
SPECIFIED

FINISH
ALL OVER

DO NOT SCALE DRAWING

MADSEN DESIGNS INC.

TITLE
MULTI-ADJUSTER SLIDE

SIZE B
CAGE CODE
DWG NO. ASSEMBLY
REV

SCALE 1:1
SHEET 1 OF 1

Index

Index

Inventor and Its Applications 2010

Inventor and Its Applications describes in detail the steps for creating sketches and models using the Autodesk Inventor® 2010 Professional software. The text teaches students how to develop and control sketches with constraints, dimensions, and tables. Once these skills are mastered, students use the sketches to build models of various complexities, add model features, create assemblies, and develop drawings. The text gives students an opportunity to show off their designs in the form of orthographic drawings and presentations.

Inventor and Its Applications contains more than 600 illustrations to reinforce the descriptions and methods covered in the chapters. This text not only teaches how to use Autodesk Inventor, but also provides skills that will help students excel in a career in drafting, design, or engineering.

Student Companion Web Site

The Student Web site for *Inventor and Its Applications* provides resources to help students get the most from the textbook.

- ✔ Exercises that allow students to practice techniques they have learned.
- ✔ Supplemental material for textbook chapters.
- ✔ Chapter tests in Microsoft® Word and PDF formats.
- ✔ Dynamic glossary that places vocabulary definitions at student fingertips.
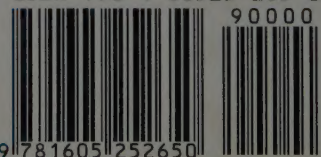- ✔ Links to many useful CAD and drafting Web sites.
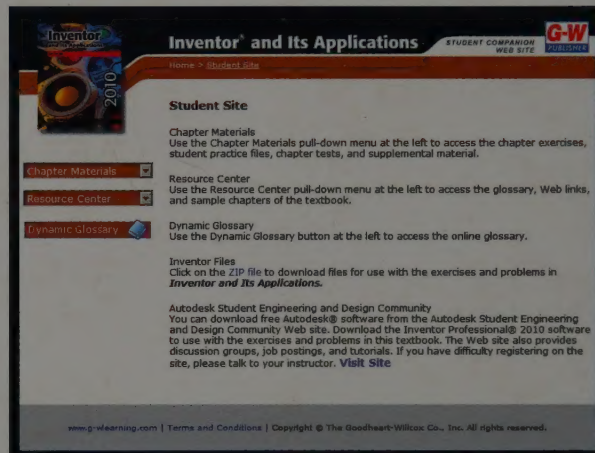
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